STUDY OF SLOSHING EFFECTS IN A CYLINDRICAL TANK WITH AND WITHOUT BAFFLES UNDER LINEAR ACCELERATION

Thesis Submitted to

National Institute of Technology, Rourkela

for the award of the degree

of

Master of Technology

In Mechanical Engineering with Specialization

in "Thermal Engineering"

by

Niraj Kumar

(Roll No. 211ME3174)

DEPARTMENT OF MECHANICAL ENGINEERING

NATIONAL INSTITUTE OF TECHNOLOGY, ROURKELA

JUNE 2013

STUDY OF SLOSHING EFFECTS IN A CYLINDRICAL TANK WITH AND WITHOUT BAFFLES UNDER LINEAR ACCELERATION

Thesis Submitted to

National Institute of Technology, Rourkela

for the award of the degree

of

Master of Technology

In Mechanical Engineering with Specialization

in "Thermal Engineering"

by

Niraj Kumar

(Roll No. 211ME3174)

Under the guidance of

Prof. A. K. Satapathy

DEPARTMENT OF MECHANICAL ENGINEERING

NATIONAL INSTITUTE OF TECHNOLOGY, ROURKELA

JUNE 2013

NATIONAL INSTITUTE OF TECHNOLOGY

ROURKELA

CERTIFICATE

This is to certify that the work in this thesis entitled "***STUDY OF SLOSHING EFFECTS IN A CYLINDRICAL TANK WITH AND WITHOUT BAFFLES UNDER LINEAR ACCELERATION***" by **Mr. Niraj Kumar** (211ME3174) has been carried out under my supervision in partial fulfilment of the requirements for the degree of **Master of Technology** in Mechanical Engineering with **Thermal Engineering** specialization during session 2011 - 2013 in the Department of Mechanical Engineering, National Institute of Technology, Rourkela.

To the best of my knowledge, this work has not been submitted to any other University/Institute for the award of any degree or diploma.

Date

Prof**. A. K. Satapathy**
Dept. of Mechanical Engineering
National Institute of Technology
Rourkela-769008

ACKNOWLEDGEMENT

I am extremely fortunate to be involved in an exciting and challenging research project like "**Study of Sloshing Effects in a Cylindrical Tank With and Without Baffles under Linear Acceleration**". It has enriched my life, giving me an opportunity to work in a new environment of ANSYS (FLUENT). This project increased my thinking and understanding capability as I started the project from scratch.

I would like to express my greatest gratitude to my supervisor **Prof. A. K. Satapathy**, for his excellent guidance, valuable suggestions and endless support. He has not only been a wonderful supervisor but also a genuine person. I consider myself extremely lucky to be able to work under guidance of such a dynamic personality. Actually he is one of such genuine person for whom my words will not be enough to express.

I would like to express my sincere thanks to **Mr. Sachindra Kumar Raut (**Ph.D. Research Scholar) and all my classmates specially **Amit Kumar Yadav** and **Hitesh Dimri** for their precious suggestions and encouragement to perform the project work. I am very much thankful to them for giving their valuable time for me.

Finally, I express my sincere gratitude to my parents for their constant encouragement and support at all phases of my life.

Date:

Niraj Kumar
Roll. No. 211ME3174
Thermal Engineering
Dept. of Mechanical Engineering
NIT Rourkela

ABSTRACT

Any motion of liquid surface is termed as a sloshing. It can be caused by any disturbance to partially filled liquid container. So to constitute a slosh dynamic of liquid, liquid must have free surface. The slosh dynamic of liquid alters the system dynamic or stability significantly when liquid is interact with the container. The motion of liquid with a free surface is of great concern in many engineering disciplines such as propellant slosh in spacecraft, rockets (especially upper stages), cargo slosh in ship and trucks transporting liquid (for example oil and gasoline) etc. To reduce the sloshing in moving container baffle is used as suppressing device. The aim of the present simulation to study the variation of force and moment caused by sloshing, when the tank is subjected to longitudinal and combined longitudinal and lateral acceleration, with transverse and transverse baffle with hole. In the present study a 3-D transient analysis of a fuel tank to acceleration field is done by using ANSYS-FLUENT v13.0. The model used in the simulation is multiphase with volume of fluid (VOF) for tracing of interface. Simulation is carried out for 20 sec by using variable time.

Keywords: Sloshing, baffle, multiphase, volume of fluid (VOF), transverse baffle, transverse baffle with holes.

CONTENTS

List of Figures

List of table

Nomenclature

English Symbols

g	Acceleration due to gravity
h	height
b	depth
a	Acceleration
$\vec{F}$	Force vector
I	Unit tensor
VOF	Volume of fluid
P, q	phases
X, Y, Z	Co-ordinate axes
t	Time
$\vec{M}$	Moment vector
Δt	Time step
Δx_{cell}	Cell size
c	Courant number

Greek Symbol

k	Turbulence kinetic energy
ε	Turbulence dissipation
α_q	q^{th} fluid volume fraction
ρ	Density
μ	Viscosity

Chapter 1

Introduction

Introduction

1.1 General

The movement of liquid inside another object is known as Sloshing. The liquid sloshing may cause various engineering problems, for example instability of ships, in aerospace engineering and ocean engineering [3], failures on structural system of the liquid container [5]. The motion of liquid with a free surface is of great concern in many engineering disciplines such as propellant slosh in spacecraft tanks and rockets (especially upper stages), cargo slosh in ship and trucks transporting liquid (for example oil and gasoline), oil oscillation in large storage tanks, water oscillation in a reservoir due to earthquake, sloshing of water in pressure-suppression pools of boiling water reactors and several others.

Depending on the types of disturbance and container geometry, the free liquid surface can experience different types of motion including planar, non-planar, rotational, and symmetric, asymmetric, disintegration, quasi-periodic and chaotic. When interacting with its elastic container or its support structures, the free liquid surface can exhibit fascinating types of motion in the form of energy exchange between interacting modes. Under the low gravity field, the surface tension is dominant and the liquid may be oriented randomly within the tank depending essentially upon the wetting characteristics of the tank wall.

The basic problem of liquid sloshing involves the estimation of hydrodynamic pressure distribution, moments, forces and natural frequencies of the free surfaces of the liquid. These parameters have a direct effect on the dynamic stability and performance of moving containers. Generally, the hydrodynamic pressure of liquids in moving rigid containers has two distinct components. One component is directly proportional to the acceleration of the tank. This component is caused by the part of the fluid moving with the same tank velocity. The second is known as convective pressure and represents the free-surface-liquid motion. Mechanical models such as mass-spring-dashpot or pendulum systems are usually used to model the sloshing part.

Liquid motion inside its container has an infinite number of natural frequencies, but it is the lowest few modes that are most likely to be excited by the motion of a vehicle. Most studies have therefore concentrated on investigating forced harmonic oscillations near the lowest natural frequencies predicted by the fluid field linear equations. However, nonlinear effects result in the frequency of maximum response being slightly different from the linear natural frequency and depend on amplitude. Nonlinear effects include amplitude jumps, chaotic liquid surface motion, parametric resonance, and nonlinear sloshing mode interaction due to the occurrence of internal resonance among the liquid sloshing modes. The nonlinearities associated with free-surface motion inside moving containers are different from those nonlinear water waves in the ocean and canals.

1.2 Linear Wave Theory

Sloshing is often analyzed in a simpler form where no overturn takes place, that is to say when the free surface strays intact. Assumptions like incompressibility, irrotational flow, inviscid (viscous/drag/friction terms are negligible), no ambient velocity (no current), two-dimensional and small amplitudes allow for a simplified analysis via linear wave theory [12].

1.3 Liquid Sloshing in moving vehicles

The problem of liquid sloshing in moving or stationary containers is of great concern to aerospace, nuclear and civil engineers, designers of road tankers [1], physicists, and ship tankers and mathematicians. Seismologists and civil engineers have been studying liquid sloshing effects on oil tanks, large dams, and elevated water towers underground motion [3]. They also mounted liquid tanks on the roofs of multistory buildings as a means to control building oscillations due to earthquakes [7].

The dynamic stability of liquefied natural gas (LNG) tankers and ship cargo tankers, and liquid hydrodynamic impact loading are problems of current interest to the designers of such systems. In populated cities, gasoline and other flammable liquid tankers are prone to rollover accidents, while entering and exiting highways. This is a very difficult mathematical problem to solve analytically, since the dynamic boundary condition at the free surface is nonlinear and the position of the free surface varies with time in a manner not known a priori. Other important contributions include the development of digital computer codes to solve complex problems that were difficult to handle in the past.Analytical solutions of sloshing in containers are limited to regular geometric tank shapes such as cylindrical, and rectangular.

Different tank geometries that are under the consideration of sloshing analysis are shown in Figures 1.1 (a)-(e).

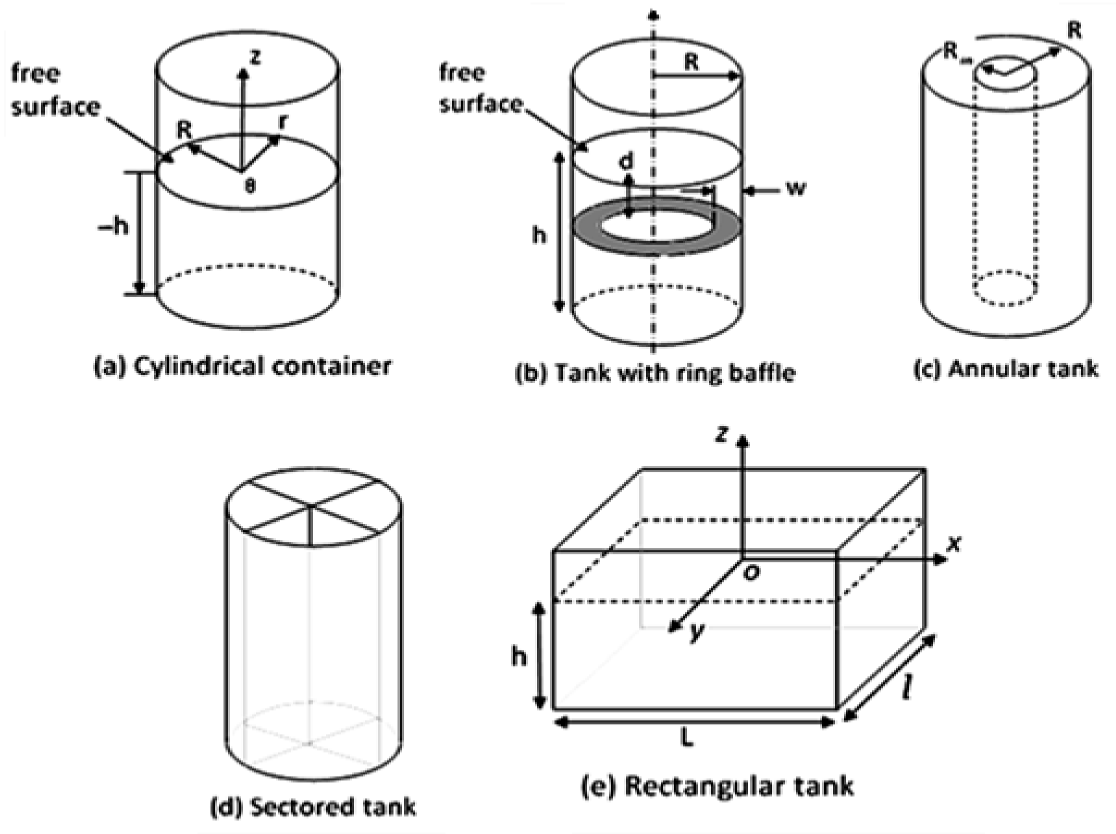


Fig 1.1: Different Tank Geometries

The nature of sloshing dynamics in cylindrical tanks is better understood than for prismatic tanks. However, analytical techniques for predicting large-amplitude sloshing are still not fully developed. In addition, much of the sloshing technology developed for space applications is not applicable to road tankers. The reason is that more emphasis has been placed on frequencies and total forces as they relate to control system requirements. Further, the excitation amplitudes considered in space applications are too small for road vehicle simulation. To avoid catastrophic sloshing in space vehicles, the control system frequencies, the vehicle elastic structure frequencies, and the fluid-slosh frequencies must be fairly widely separated.

Sloshing phenomena in moving rectangular tanks can usually be described by considering only two-dimensional fluid flow if the tank width is much smaller than its breadth. Sloshing in spherical or cylindrical tanks, however, is usually described by three-dimensional flow. Tanks with two-dimensional flow are divided into two classes: low and high liquid fill depths. The low fill depth case is characterized by the formation of hydraulic jumps and traveling waves for excitation periods around resonance. At higher fill depths, large standing waves are usually formed in the resonance frequency range. When hydraulic jumps or traveling waves are present, extremely high impact pressures can occur on the tank walls. Impact pressures are only measured experimentally and cannot be estimated theoretically or numerically.

1.4 Suppression Devices

As the size of liquid containers increases the hydrodynamic forces and moments become very large particularly in the neighborhood of resonance. In an attempt to avoid structural failure or undesirable dynamic behavior, one must introduce some means to suppress or to reduce the sloshing dynamic loads. This may be achieved by introducing baffles of different configurations. Such baffles change the fluid natural frequencies depending on the shape, size and position of baffles. Anti-slosh baffles are expected to prevent the total liquid force from exceeding a certain prescribed maximum value under all possible filling conditions, tank orientation, and external excitation.

The primary parameters influencing the design of slosh-suppression devices [2] are as follows:

1. The vehicle's mission profile and trajectory.
2. The damping requirements for a given container, or liquid-slosh-motion amplitudes at various liquid levels.
3. The physical characteristics of the tank such as its geometry, elastic deformation, and insulation.
4. The liquid filling and draining requirements.
5. The physical properties of the liquid.
6. The handling, slosh and impact loads that must be sustained by the devices.

Fig1.2 shows some common types of baffles and anti-slosh devices. The basic classification of these baffles is as follows:

1. Horizontal baffle rings, which can be movable, or fixed, or rings with radial clearance as shown in Fig 1.2 (a)-(c).
2. Conical baffles, which are placed upright/inverted as shown in Fig 1.2 (d) and (e)

3. Radial (sectored) baffles or cruciform in the form of complete sectored baffles as shown in Fig 1.2 (f)-(i).
4. Annular tank.
5. Z-ring conic section baffles.
6. Floating cans
7. Floating lid devices.

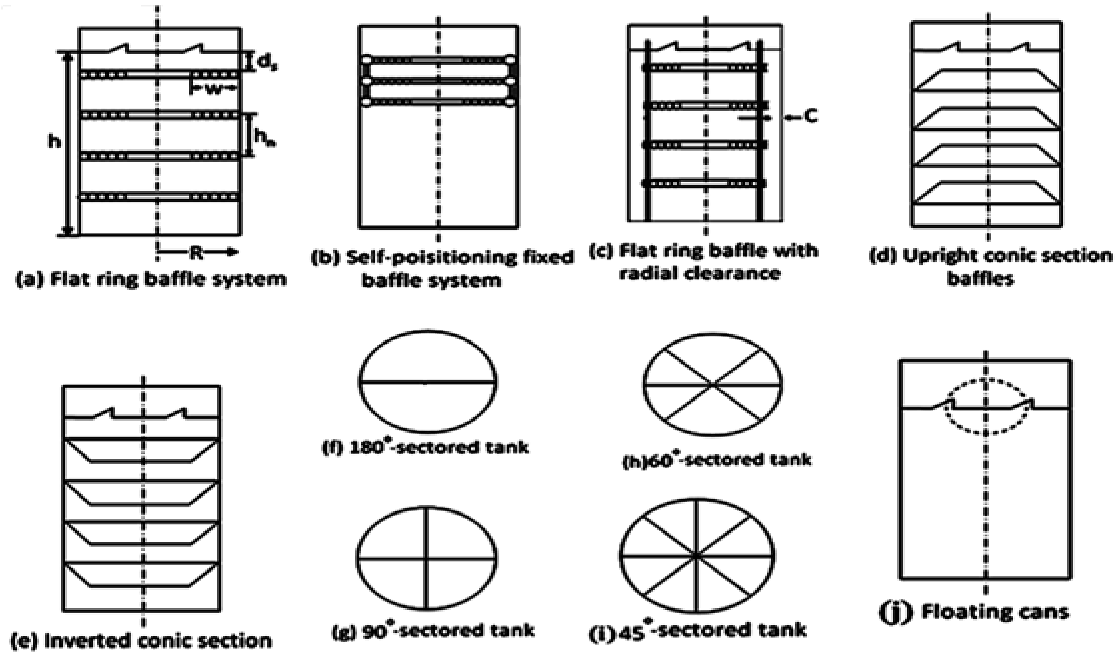


Fig 1.2: Sloshing Suppression Devices

1.5 Free Surface Representation

Several techniques exist for tracking immiscible interfaces, each with its own strengths and weaknesses (tracking). These techniques can be classified under three main categories according to physical and mathematical approaches:

1. Capturing (also known as moving grid or Lagrangian approach).
2. Tracking (also known as fixed grid or Eulerian approach).
3. As well as combined methods of both 1 and 2.

Capturing methods include moving-mesh, particle-particle scheme, and boundary integral method. Tracking methods can be divided into two main approaches: surface tracking and volume tracking. These include front-tracking, volume-of–fluid (VOF) [8], marker and cell

(MAC) method, smoothed particle hydrodynamics, level set methods, and phase field. Combined methods include the mesh free/particle method, Coupled Eulerian-Lagrangian and variants from the previously mentioned two methods. Amongst these, an indicator function is used which is a volume fraction (color function) for VOF methods or a level set for level-set methods.

1.6 Computational Fluid Dynamics Packages

The advent of modern computing capabilities, their dramatic reduction in cost and their widespread integration into the modern engineering design office have changed the way the design process is approached by the modern engineer. The availability of affordable high performance computing hardware and the introduction of user-friendly interfaces have lead to the development of commercial CFD packages. Several general-purpose CFD packages have been published in past decade. Prominent among them are: PHONICS, FLUENT, SRAT-CD, CFX, FLOW-3D and COMPACT. Most of them are based on the finite volume method. Due to the availability, the commercial CFD code **fluent v13.0** is used throughout the study for the modeling of the sloshing event.

1.7 Objective of the present work

- To study the concept of liquid sloshing.
- To simulate fuel sloshing in a cylindrical tank subjected to longitudinal, and combined longitudinal and lateral acceleration at different fill level by using ANSYS -FLUENT v13.0 software.
- To study the effect of slosh forces and moment on stability and braking performance of vehicles.
- To study the effectiveness of transverse baffle and transverse baffle with holes for reducing slosh forces and moment.

1.8 Thesis Organization

This thesis has been organized in total seven chapters.

Chapter 1 presents a brief introduction to liquid sloshing and its effects on system dynamics. The suppression devices like baffles are discussed along with aim of the present investigation.

Chapter 2 presents an extensive survey of available literature on various aspects of liquid sloshing.

Chapter 3 presents the physical model and problem formulation of the present study.

Chapter 4 in chapter 4 mathematical modeling of the present study has been carried out by discussing governing equation for multiphase model.

Chapter 5 describes the computational fluid dynamics and set-up of ANSYS-FLUENT of present study.

Chapter 6 discussed about the result of the present study.

Chapter 7 is confined to some concluding remarks and for outlining the scope of future work.

1.9 Closure.

In this chapter we know about sloshing. By introducing baffles of different configurations one can suppress or reduce the sloshing dynamic loads. Modern engineer is gifted with many commercial CFD packages enabling him to handle such complex problems like sloshing.

Chapter 2

Literature Review

Literature Review

2.1 Introduction

In the current chapter, the summary of the literature surveyed during the course of this research has been presented. This survey of literature is expected to provide the background information and thus to select the objectives of the present investigation on sloshing.

2.2 Importance of sloshing

Sloshing in tanks has received much attention over the years since the mid-1950s (Graham and Rodriguea 1952). The major field of interest was initially aeronautics, where the motion of fuel in the fuel tanks would affect the dynamics of a plane. Fuel in space rockets became a further field of focus. More recently, the motion of liquids, including fuel, in numerous maritime applications and its structural and destabilizing effects attracted much attention. Further fields of interest include the aerodynamic and seismic stabilization of tall structures and the acoustical effects of fuel sloshing in vehicle fuel tanks.

2.3 Computational studies

K.M.Tehrani et al. [1] studied sloshing of fuel in a partly filled cylindrical tank equipped with baffle. In this study, three-dimensional transient analysis are performed under varying magnitude of constant longitudinal, lateral and combination of longitudinal and lateral acceleration of tank, and two different fill volume using FLUENT software. The result of investigation are presented in term of amplification factor of slosh forces and moment and variation in the mass moment of inertia of fuel within a clean bore tank and a baffled tank, for two different magnitude of acceleration excitation and two different fill volume. The result of above study clearly shows that amplification factor of forces and moment for clean bored tank is about 2 and amplification factor reduces significantly by inserting transverse baffle.

S. Rakheja et al. [2] done an analysis of effectiveness of different design of baffle, including lateral, conventional, partial and oblique under combined constant longitudinal and lateral acceleration for different fill level. The analysis done by using FLUENT software with volume of fluid (VOF) model for tracing of interface between two immiscible fluids. The result shows that the conventional lateral baffle offers strong resistance to fluid slosh under longitudinal acceleration only while the oblique baffle helps to reduce longitudinal as well as lateral slosh forces and moment under combined longitudinal and lateral acceleration of the fuel tank.

Ibrahim Raouf A et al. [3] deals with the theory of linear liquid sloshing dynamics. Addresses the nonlinear theory of liquid sloshing dynamics, Faraday waves, and sloshing impacts; presents the problem of linear and nonlinear interaction of liquid sloshing dynamics with elastic containers and supported structures.

Kingsley et al. [4] have studied about design and optimization of container for sloshing and impact. They have used numerical simulation and experimental validation. In numerical technique they used a rectangular tank for simulation and further volume of fluid (VOF) method for tacking the free surface, k-Ɛ turbulence model for viscous effect and an acceleration curve imposed as userdefined function (UDF). Since the problem has no inlet and outlet boundaries.

D.Takabatake et al. [5] has studied about many structural damage caused by liquid sloshing. They observed many petroleum tanks were damaged by sloshing during 2003Tokachi-oki, Japan. Large earthquakes are predicted to occur within 50 years, which will cause the similar damage. They developed a splitting wall as a new sloshing reduction device. Model experiments and numerical simulations are performed to investigate the effect of the device. The experimental results indicate the proposed device is effective to reduce sloshing against sinusoidal input motion. They also perform numerical study to simulate the experimental result. Both experimental and numerical results are almost same. Based on the numerical simulation, the proposed device can be also effective against earthquake ground motion.

Eswaran et al. [6] Studied the effect of baffles on liquid sloshing for the partially filled cubic tank. Numerical simulations were carried out by using volume of fluid technique with arbitrary Langragian-Eulerian formulation. The response of the couple system is obtained by using well known software ADINA.

2.4 Numerical and Experimental studies

Wei Chen et al. [7] simulated large-amplitude sloshing motion of liquid subjected to harmonic and earthquake base excitations. It is concluded that non-linear sloshing effects should be considered in seismic-resistant tank design. It would take into account the true sloshing amplitudes which could be in excess of the linear predictions to avoid the damage on the upper part of the structure incurred by the pounding of the excessive hydrodynamic waves. The linear theory was found to be generally non-conservative in the sloshing amplitude but was reasonably accurate in predicting peak hydrodynamic forces.

M. H. Djavareshkian et al. [8] use the one of the most popular Finite volume methods called VOF (Volume of Fluid) method for tracking the flow in containers. They open a new horizon on the simulation of sloshing phenomena in their journal.

Sakai et al. [9] investigated the sloshing behavior of floating- roofed oil storage tanks through theoretical analysis and model testing. The analysis employed theory of fluid-elastic vibration to study the interaction between a roof and the contained liquid. The theory was verified by shake table experiment with three large models of single deck type and double deck type of floating roofs.

Pal et al. [10] used the finite element technique to study the dynamics of inviscid, incompressible liquid inside thin walled, flexible, composite cylindrical tanks under small displacements in a coupled manner. The finite element equation of motion for both the tank wall and the fluid domain was formulated. Both rigid and flexible tank systems were analyzed to demonstrate the effect of tank flexibility on the slosh characteristics and the structural response. An experimental set-up was made to study sloshing frequencies, sloshing displacements and hydrodynamic pressure. It was found that liquid slosh frequencies in rigid container decrease in liquid depth and increase in container width.

Abramson et al. [11] has applied linear theories, based on the potential formulation of velocity field, to the analysis of the liquid motion in cylindrical and spherical tanks and in ring and circular sector compartmented tanks. Experimental tests have been carried out in order to validate the mathematical models and to understand the effect of the geometrical and physical variables on the free surface oscillation.

Hasheminejad et al. [12] introduced a two dimensional hydrodynamic analysis based on the linear potential theory to study the natural sloshing frequencies of transverse mode in a half filled non deformable horizontal cylindrical container of elliptical cross section, without or with a pair of inflexible horizontal longitudinal side baffles of arbitrary extension positioned at the free liquid surface.

Biswal et al. [13] studied the influence of annular baffle on the dynamic response of a partially liquid-filled cylindrical tank. Baffles are generally used as damping devices in liquid storage tanks. The focus of their work is to study the influence of a baffle on the dynamic response of a partially liquid-filled cylindrical tank. A baffle is assumed here to have the shape of a thin annular circular plate.

Celebi et al. [14] have simulated the problem of fluid motion in partially filled rectangular tanks using the volume of fluid formulation to track the free surface. They solved the complete Navier-Stokes equation in primitive variables by the use of the finite difference approximations. The study also included the use of a vertical baffle to have a pronounced effect in shallow water.

Rebouillat et al. [15] described recent studies devoted to the problem of modeling the fluid structure interaction in partially filled liquid containers. The problem of fluid structure interaction is of much importance for numerous applications, including those in the aerospace industry (rockets, satellites), naval and road transportation, and has become in recent years an object of keen interest from researchers. They focused in particular on the sloshing phenomenon and on the numerical approaches used to predict the sloshing, frequency, wave amplitude and pressure exerted on the walls and the effect of sloshing on the stability in the container environment.

2.5 Closure:

This chapter describes the various research works about sloshing. Many research papers were published regarding liquid sloshing. Some of them investigated computationally, some are numerically and experimentally predicted. Sloshing is a real and present challenge due its nonlinear behavior, scientific approach in analyzing liquid sloshing is essential in developing anti sloshing devices. Mostly analytical solutions are newly developed techniques which needs validation. So the experimental results are also required to validate the CFD codes and analytical results. In this study we solved the governing equations by volume of fluid (VOF) method and analyze the results which are obtained.

****

Chapter 3

Physical Model

Physical Model

3.1 Physical Model

The physical model of the present study is shown in Fig3.1 The physical model consist a cylindrical fuel tank of a moving vehicle, which is partially filled with gasoline (ρ = 850 kg/m^3 , μ = 0.0687 kg/m-s) [2]. The dimensions of the tank are of 8 m length and 2m diameter [1]. The gasoline occupies either 40% or 80% of the total volume of the space in the cylinder. The remaining part of the tank is occupied with air. In the accelerated and decelerated phases of the vehicle the liquid fuel is set in sloshing mode. Even in the uniform motion (cruising phase), minor disturbing forces also contribute for sloshing. However, in this study we considered only the sloshing due to the acceleration and deceleration of the vehicle .To reduce the sloshing rate three similar circular baffles are introduced in the tank. The spacing between adjacent baffles are equal and, the centers of the baffles coincide with the axis of the tank. The details various dimensions of the physical model are given in Fig 3.1.

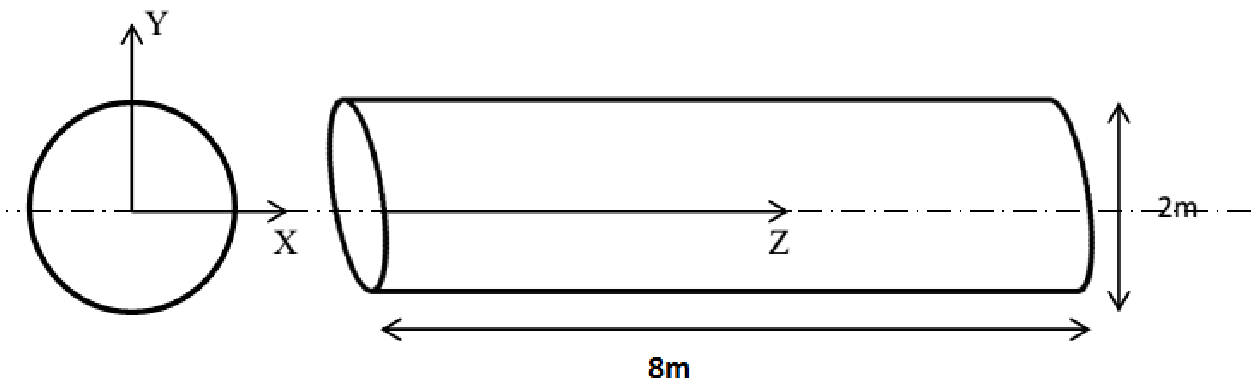


Fig 3.1: Geometry of the fuel tank

3.2 Motion of Tank

The study has been done for two different cases of acceleration direction and magnitude:

i. When the fuel tank is subjected to only longitudinal acceleration and deceleration for two different magnitudes of a_z = 3.47 m/s^2 and a_z = 4.17 m/s^2.

ii. When the fuel tank is subjected to combined longitudinal and lateral acceleration and deceleration for two different cases of magnitude:

1. $a_z = 3.47\ \text{m/s}^2$ and $a_z = 1.5\ \text{m/s}^2$
2. $a_z = 4.17\ \text{m/s}^2$ and $a_z = 1.5\ \text{m/s}^2$

3.2 Linear Motion of the tank

The moment of the rectangular tank is described in the Fig3.2. The fuel tank is subjected to a constant acceleration from 0 to 100 km/h in 8 seconds and then left to move with a constant velocity of 100 km/h for 4 seconds. Then the tank was decelerated to rest in 8 seconds. This seems to be an acceptable motion of a vehicle. Hence all the simulation works are carried out for at least 20 seconds.

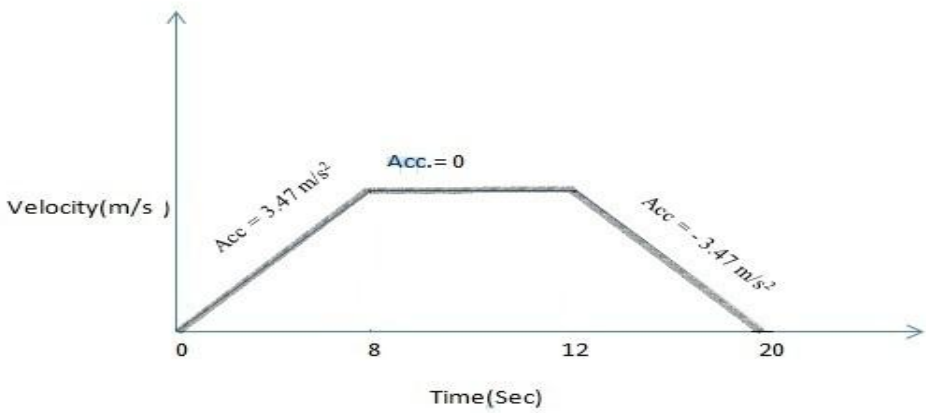


Fig 3.2: Tank Motion

3.3 Types of baffle used

Sloshing occurs is due to accelerating and decelerating motion of a partially filled tank, so that to reduce the sloshing and its adverse effect baffle is placed inside the cylinder. In the present study, simulation is done with and without baffle and studied the effectiveness of different orientation of the baffle on sloshing forces and moment. The spacing between adjacent baffles are equal and, the centers of the baffles coincide with the axis of the tank.

The different orientation baffle is studied are as follows:

1. Three equi-spaced transverse baffle of width 0.4m

2. Three equi-spaced transverse baffles (width 0.4m) with holes of diameter of 0.15m.

The baffle is annular in cross-section with 1.2m inner diameter and 2m outer. The geometry is shown below:

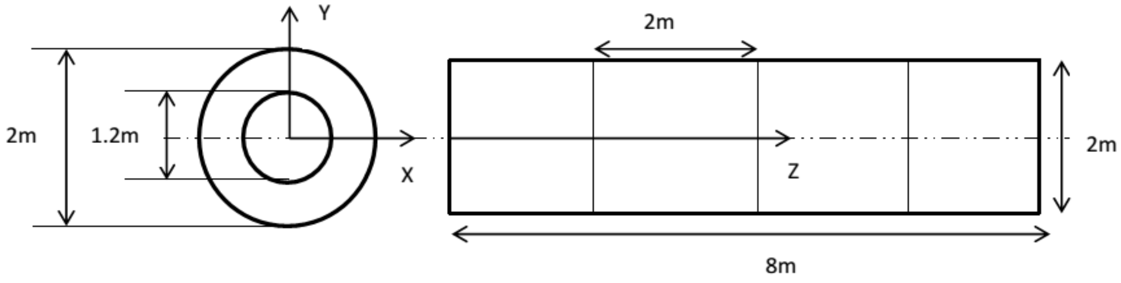


Fig 3.3 Cylinder with transverse baffles

3.4 Closure:

Physical model of the present study, Motion of tank and different types of baffle with geometry explained in this section.

Chapter 4

Mathematical Model

Mathematical Model

A mathematical model is a type of formal interpretation (also called an abstract model) that uses mathematical language to describe a system. Eykhoff (1974) defined a mathematical model as 'a representation of the essential aspects of an existing system (or a system to be constructed) which presents knowledge of that system in usable form.

This chapter gives a brief explanation of the equations solved by Computational Fluid Dynamics (CFD) codes and detail explanation of capturing the free surface using VOF technique.

4.1 Governing Equation

4.1.1 Continuity Equation

A continuity equation describes the transport of a conserved quantity. Continuity equation also defined as conservation of mass. The law of conservation of mass state that mass can neither be created nor be destroyed. Three dimensional, incompressible, unsteady, continuity equations is as follows:

$$\frac{1}{r}\frac{\partial}{\partial r}\left(r u_r\right)+\frac{1}{r}\frac{\partial u_\phi}{\partial \phi}+\frac{\partial u_z}{\partial z}=0 \tag{4.1}$$

Where u_r, u_ϕ and u_z are velocity component in r, ϕ and z direction.

4.1.2 Nevier-stokes equation (Momentum equation)

The Navier-Stokes equations are the basic governing equation for viscous, heat conducting fluid.it is a vector equation obtained by applying newton's law of motion to a fluid element and is also called the momentum equation.

Navier-stoke equation considers the viscosity of fluid because in most cases we cannot neglect the viscous effect.

r- Momentum equation

$$\rho\left(\frac{\partial u_r}{\partial t}+u_r\frac{\partial u_r}{\partial r}+\frac{u_\phi}{r}\frac{\partial u_r}{\partial \phi}+u_z\frac{\partial u_r}{\partial z}-\frac{u_\phi^2}{r}\right)=-\frac{\partial p}{\partial r}+\mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_r}{\partial r}\right)+\frac{1}{r^2}\frac{\partial^2 u_r}{\partial \phi^2}+\frac{\partial^2 u_r}{\partial z^2}-\frac{u_r}{r^2}-\frac{2}{r^2}\frac{\partial u_\phi}{\partial \phi}\right]+\rho g_r \tag{4.2}$$

ϕ- Momentum equation

$$\rho\left(\frac{\partial u_\phi}{\partial t}+u_r\frac{\partial u_\phi}{\partial r}+\frac{u_\phi}{r}\frac{\partial u_\phi}{\partial \phi}+u_z\frac{\partial u_\phi}{\partial z}+\frac{u_r u_\phi}{r}\right)=-\frac{1}{r}\frac{\partial p}{\partial \phi}+\mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_\phi}{\partial r}\right)+\frac{1}{r^2}\frac{\partial^2 u_\phi}{\partial \phi^2}+\frac{\partial^2 u_\phi}{\partial z^2}+\frac{2}{r^2}\frac{\partial u_r}{\partial \phi}-\frac{u_\phi}{r^2}\right]+\rho g_\phi \tag{4.3}$$

z- Momentum equation

$$\rho\left(\frac{\partial u_z}{\partial t}+u_r\frac{\partial u_z}{\partial r}+\frac{u_\phi}{r}\frac{\partial u_z}{\partial \phi}+u_z\frac{\partial u_z}{\partial z}\right)=-\frac{\partial p}{\partial z}+\mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_z}{\partial r}\right)+\frac{1}{r^2}\frac{\partial^2 u_z}{\partial \phi^2}+\frac{\partial^2 u_z}{\partial z^2}\right]+\rho g_z \tag{4.4}$$

Where p is the static pressure, u_r, u_ϕ and u_z are the velocity components in r, ϕ, and z direction. μ is the dynamic viscosity, ρ is density and ρg_r, ρg_ϕ and ρg_z are body force due to gravity.

4.2 Multiphase governing equations

4.2.1 Conservation of momentum [18]

A single momentum equation is solved throughout the domain, and the resulting velocity field is shared among the phases. The momentum Equation, shown below, is dependent on the volume fractions of all phases through the properties ρ and μ.

$$\frac{\partial}{\partial t}\left(\rho\vec{V}\right)+\nabla.\left(\rho\vec{V}\vec{V}\right)=-\nabla p+\nabla.\left(\tau\right)+\rho g+F \tag{4.5}$$

Where p is the static pressure, $\vec{\tau}$ is the stress tensor and $\rho\vec{g}$, $\vec{F}$ are the gravitational body force and external body force (e.g., that arise from interaction with the dispersed phase), respectively. $\vec{F}$ Also contains other model-dependent source terms such as porous-media and behaves much like the gravitational body force, but allows for user-defined source terms, here momentum source. Momentum source has the units of

Kg/m^2s^2 and is the multiple of the density of a specific mesh cell and the instantaneous acceleration. This term will prove very useful in the implementation of load curves in this study.

4.2.2 Volume of fluid model [18]

A further model that needs to be discussed, due to its relevance to this study, is the multiphase volume of fluids method (VOF). The model was originally developed by Hirt, et al. [22]. In computational fluid dynamics, the volume of fluid (VOF) method is a numerical technique for tracking and locating the free surface (or fluid-fluid interface). The volume of fluid method is based on earlier Marker-and-cell (MAC) methods. First accounts of what is now known as VOF have been given by Noh & Woodward (1976). VOF model solve phase and the total continuity equation and the result is pressure and volume fraction which points out where the interface is. Applications of the VOF model include stratified flows, filling, free-surface flows, sloshing, the motion of large bubbles in a liquid, the motion of liquid after a dam break, the prediction of jet breakup (surface tension), and the steady or transient tracking of any liquid-gas interface.

4.2.3 Volume Fraction [18]

The VOF formulation relies on the fact that two or more fluids (or phases) are not interpenetrating. For each additional phase that you add to your model, a variable is introduced: the volume fraction of the phase in the computational cell. In each control volume, the volume fraction of all phases sums to unity. The field for all variable and properties are shared by the phases and represent volume averaged value, as long as the volume fraction of each of the phases is known at each location. Thus the variable and other words, if the q^{th} fluid's volume fraction in the cell is denoted as α_q, then the following three conditions are possible:

- $\alpha_q = 0$: the cell is empty (of the q^{th} fluid).
- $\alpha_q = 1$: the cell is full (of the q^{th} fluid).
- $0 < \alpha_q < 1$: the cells contain the interface between the qth fluid and one or more other fluids

Based on the local value of α_q, the appropriate properties and variable will be assigned to each control volume within domain.

4.2.4 Illustration of VOF Model

VOF model can be well illustrated using the following 2D rectangular tank (see Figure3.4), in which the dark area represents water and the remaining portion represents air. This is taken from the simulation of sloshing on a rectangular tank at an instant.

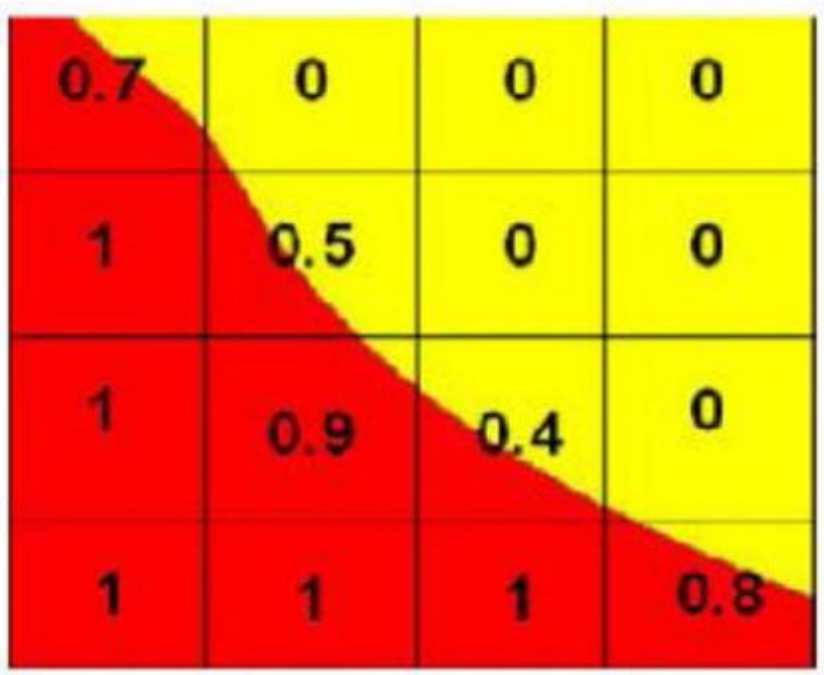


Fig 3.4 Volume of Fluid

The values that are marked in the cells represent their corresponding volume of fraction of water. In the absence of water the cell value is zero ($\alpha_{water} = 0$) and in the absence of air this has become unity ($\alpha_{water} = 1$). If the cell value is intermediate between the zero and unity say $\alpha_{water} = 0.4$, then it represents that the cell contains an interface between the two fluids air and water. [18]

4.2.5 Dynamic sloshing forces

The magnitudes of forces in the steady and transient state are derived from the pressure distribution over the wetted boundaries of the baffled as well as the clean bore tanks. The resultant slosh force are computed from the distributed pressure through integration over the wetted area of the wall cell, such that

$$F_x = \sum_{c}^{wetarea} P_c \overrightarrow{A_c} . \vec{i} \tag{4.6}$$

$$F_y = \sum_{c}^{wetarea} P_c \overrightarrow{A_c} . \vec{j} \tag{4.7}$$

$$F_z = \sum_{c}^{wetarea} P_c \overrightarrow{A_c} . \vec{k} \tag{4.8}$$

Where F_x, F_y, F_z, are the resultant slosh forces acting on the tank wall along the fixed x, y and z axes due to pressure P_c acting on cell "c" with area vector $\overrightarrow{A_c}$. $\vec{i}$, $\vec{j}$ and $\vec{k}$ athe unit vector in the x, y and z direction respectively. [1]

4.2.6 Sloshing moment

Apart from the dynamic slosh forces, the moment caused by variation in the cg coordinate are known to be significant factor in the view of the directional response of the vehicle. The roll, pitch and yaw moment about point "o" located at the bottom of the tank directly beneath of origin are obtained upon integrating the moment corresponding to each cell over the wetted area:

$$\overrightarrow{M} = \sum_{c}^{wetarea} \bar{r}_c \times F_c \tag{4.9}$$

Where $\overrightarrow{F_c}$ is the force vector caused by a cell "c" on the boundary, $\vec{r_c}$ is the position vector of cell "c" with respect to "o" and $\overrightarrow{M}$ is the moment vector about point "o". The coordinate of this point "o" are (0,-R, 0), where R is the tank radius. [1]

4.2.7 Turbulence Modelling

This study utilizes Eddy viscosity Models (EVM) to close the Reynolds-Averaged Navier-Stokes Equations (RANSE). An example of this is the k- ε turbulence model that assumes proportionality between Reynolds stresses in the fluid and mean velocity gradients. The standard k-ε model is a semi-empirical model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε).

The model transport equation for k is derived from the exact equation, while the model transport equation for ε was obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart. In the derivation of the k-ε model, it was assumed that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The standard k-ε model is therefore valid only for fully turbulent flows.

Although the form of the momentum equations remains the same, the viscosity term becomes an effective viscosity μ_{eff} , and is determined by the sum of the molecular viscosity μ and a turbulent viscosity μ_t . The turbulent (or eddy) viscosity μ_t is computed by combining k and ε as follows:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{4.10}$$

Where C μ is an empirically derived proportionality constant, whose default value can be determined from experiments with air and water for fundamental turbulent shear flow including homogeneous shear flow and decaying isotropic grid turbulence. This equals 0.09 in Fluent. This has been found to work fairly well for a wide range of wall-bounded and free shear flows.

For the present simulation this k- ε turbulence model is chosen, as it is playing a vital part in solving problems with VOF technique even today [8].

4.3 Closure

This chapter gives a brief explanation of the equations solved by Computational Fluid Dynamics (CFD) codes. A detailed explanation of continuity equation, Navier -Stoke equation and capturing the free surface using VOF technique, are discussed mathematically. The term of dynamic sloshing forces and moment is also explained mathematically.

Chapter 5

Computational Analysis

Computational Analysis

This chapter contains an introduction to computational fluid dynamics and its application in industry. The chapter includes a brief description of discretization technique and flow process of CFD simulation. This chapter also includes ANSYS Workbench and FLUENT setup of present study as well grid independent test.

5.1 Background

During the last three decades, Computational Fluid Dynamics has been an emerged as an important element in professional engineering practice, cutting across several branches of engineering disciplines. Computational fluid dynamics (CFD) is the branch of heat transfer and fluid mechanics that uses algorithm code and numerical method to analyze problem that involve fluid flow by means of computer based simulation.

5.2 Computational Fluid Dynamics

Computational fluid dynamics is the science of predicting fluid flow, chemical reactions, heat transfer, and related phenomena by solving numerically the set of governing mathematical equations[18]:

- Conservation of mass
- Conservation of momentum
- Conservation of energy
- Conservation of species
- Effect of body forces

As we know the fluid flow and heat transfer are governed by basic fundamental equations which are in partial differential (mostly in non-linear) form so analysis or prediction of fluid flow process can be done in following ways effectively:

- Experimental Approach: In an experimental approach, analysis is done in prototype. The most reliable result about any physical process is often given by actual measurement. An experimental approach involving full-scale equipment can be used to predict how identical copies of equipment would perform under the same condition. But the cost of the experimental setup in most of cases is quite high and expensive and often impossible. Despite of the many drawbacks like high cost, time, feasibility and extrapolation of result, it has one big advantage that experimental approach is capable of producing the most realistic result.

- Analytical Approach: In analytical approach governing equation of the model is solved mathematically by using boundary condition. But there is little hope of predicting many phenomena of practical interest by using classical mathematical method of solving partial differential equation. The agreement of the analytical result with experimental is about $\pm 35^{\circ}$. The big advantage of analytical approach is that "clean" general information can be obtained. This approach is quite useful in preliminary design work. The limitation of analytical approach is the complexity of governing equation and complexity of the geometry.

- CFD or Numerical Approach: The drawbacks of experimental and analytical approach are encountered by using numerical method or CFD. In CFD the non-linear partial differential equation is solved by numerical techniques. In this approach the non-linear partial differential equation are discretized into linear algebraic form of equation over a control volume by finite difference, finite volume and finite element methods. Then set of linear algebraic equation are solved iteratively by numerical technique such as gauss sidle method and TDMA method. CFD analysis complements testing and experimentation by reducing total effort and cost required for experimentation and data acquisition.

  The advantages of CFD are it can solve non-linear P.D.E., complicated physical phenomena can be treated, time evaluation of flow can be obtained and we can simulate the ideal conditions. There are no such drawbacks in CFD approach but there are some errors in this process such as truncation error, round-off error, and machine error etc. which affect the result to some extent.

As already explained that, in computational fluid dynamics the non-linear partial differential are discretized into linear algebraic set of equation. Discretization is the method of approximating the differential equation by a system of algebraic equation for the variables at some set discrete location in space and time. The discrete locations are grid, mesh point or cell. By doing discretization continuous information from the exact solution of PDE is replaced with discrete value. Discretization is done by using finite difference, finite volume and finite element methods. The brief introduction to above method is given below:

- Finite difference method: FDM is the oldest of the three methods. This technique published as early as 1910 by L.F. Richardson. In this method domain is discretized into series of grid point i.e. Structured i, j, k, grid is required. The non-linear partial differential equation discretized by Taylor series of expansion into linear algebraic equation.

- Finite volume method: This is the "classical" or standard approach used most often in commercial software and research codes. In FVM method, the partial differential equation is discretized over a control volume by using gauss divergence theorem. Control volume is obtained by dividing the domain of interest and discretized the governing P.D.E over each control volume which results in a set of algebraic equation which can be solved iteratively. In this approach the value of variable stored at center of control volume. FVM enjoys an advantage in memory use and speed for very large problems, higher speed flows, turbulent flows, and source term dominated flows (like combustion).

  Advantage: Basic FV control volume balance does not limit cell shape; mass, momentum, energy conserved even on coarse grids; efficient, iterative solvers well developed.

  Disadvantages: false diffusion when simple numeric is used.

- Finite element method: This method is popular for structural analysis of solids, but is also applicable to fluids. The FEM formulation requires, however, special care to ensure a conservative solution. The FEM formulation has been adapted for use with the Navier-Stokes equations. In this method, a weighted residual equation is formed:

$$R_i = \iiint W_i Q dV^e$$

Where R_i, is the equation residual at an element vertex i, Q is the conservation equation expressed on an element basis, W_i is the weight factor and dV^e is the volume of the element. Advantages: highest accuracy on coarse grids. Excellent for diffusion dominated problems (viscous flow) and viscous, free surface problems. Disadvantages: slow for large problems and not well suited for turbulent flow.

5.3 ANSYS-FLUENT CFD PACKAGE

ANSYS Fluent software consists of broad physical modeling capabilities needed to model flow, heat transfer, turbulence, and reactions for industrial applications. Special models that give the software the ability to model aero acoustics, in-cylinder combustion, turbo machinery, and multiphase systems have served to broaden its reach.

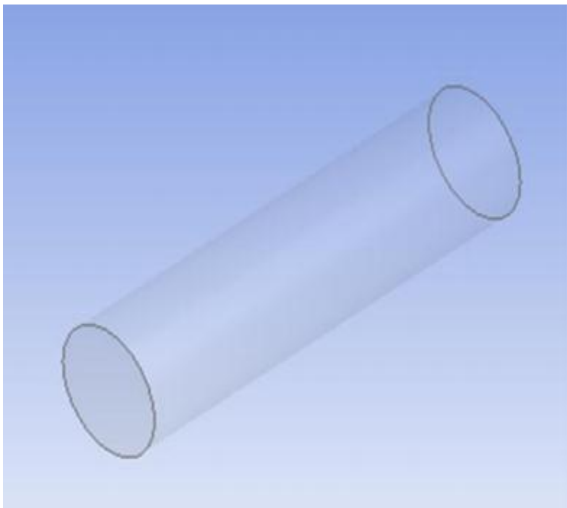
Due to mesh flexibility offered by Fluent, flow problems with unstructured meshes can be solved easily. Fluent supports triangular, quadrilateral meshes in 2D-problems and tetrahedral, hexahedral, wedge shaped meshes in 3D. The integrating of ANSYS Fluent into ANSYS Workbench provides users with superior bidirectional connections to all major CAD systems, powerful geometry modification and creation with ANSYS Design modeler technology, and advanced meshing technologies in ANSYS Meshing. By easy drag-and-drop transfer, the platform also allows data and results to be shared between applications.

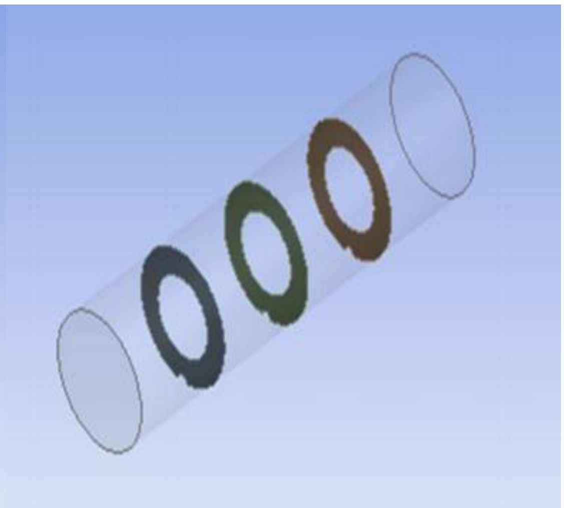
Fluent has been developed in C language and hence, provides full flexibility and power offered by the language. C language provides facilities like efficient data structures, flexible solver controls and dynamic memory allocation also. Client- server architecture in Fluent allows it to run as separate simultaneous processes on client desktop workstations and computer servers. This allows efficient execution, interactive control and complete flexibility of the machine.

5.4 ANSYS-FLUENT SETUP

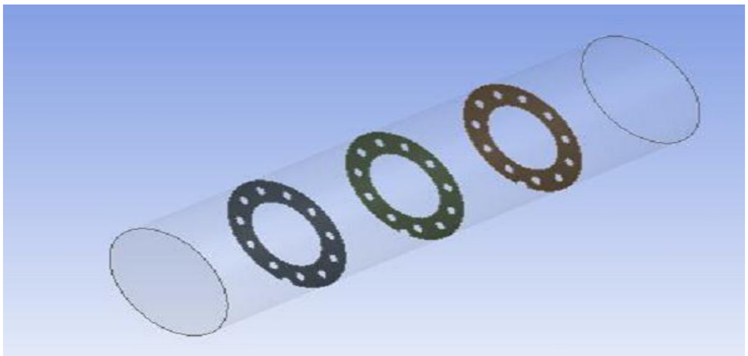
In Present study Simulation of fuel sloshing in tanker is done by using ANSYS-FLUENT v13.0. In ANSYS, workbench platform is available for modeling geometry and meshing. Workbench mesh is done automatically depending upon geometry. Then export the mesh file to a FLUENT solver and Post-processing is done. The Sequence of problem Set-Up is explained as follows:

- **Geometric Modeling:** A 3D cylinder having dimension of 8m length and 2m diameter is drawn in ANSYS DESIGN MODULAR. In design modular cylindrical geometry of without baffle, with 3 transverse baffles and 3 transverse baffles with holes are drawn. The width of baffle is 0.4m and modeled as surface in design modular. The geometry of present study is shown below.





Without baffle Transverse baffle



Transverse baffle with holes

Fig 5.1: Modeling of baffle in ANSYS

- **Mesh generation**: After drawing geometry in design modular, next meshing is done in workbench meshing tool. In this automatic meshing is done over geometry. In the present study patch independent tetrahedron mesh is generated for all cases. Simulation of transient sloshing problem is difficult because here explicit formulation for volume of fluid method used and as we know explicit formulations are conditionally stable. Stability condition is given by global courant number. During the simulation this global courant number should not be exceed by 250. The Global courant number depends on the mesh size, velocity field, and the time step size used for the transport equations. It is given by

$$c = \frac{\Delta t}{\Delta x_{cell} / v_{fluid}}$$

Where, Δt is time step, Δx_{cell} is cell distance and v_{fluid} is velocity fluid at cell. So to avoid the simulation from divergence the quality of mesh should be good one. The orthogonal quality should be greater than 0.4, skewness should be less than 0.75. The meshing of present study for different cases as shown in figure:

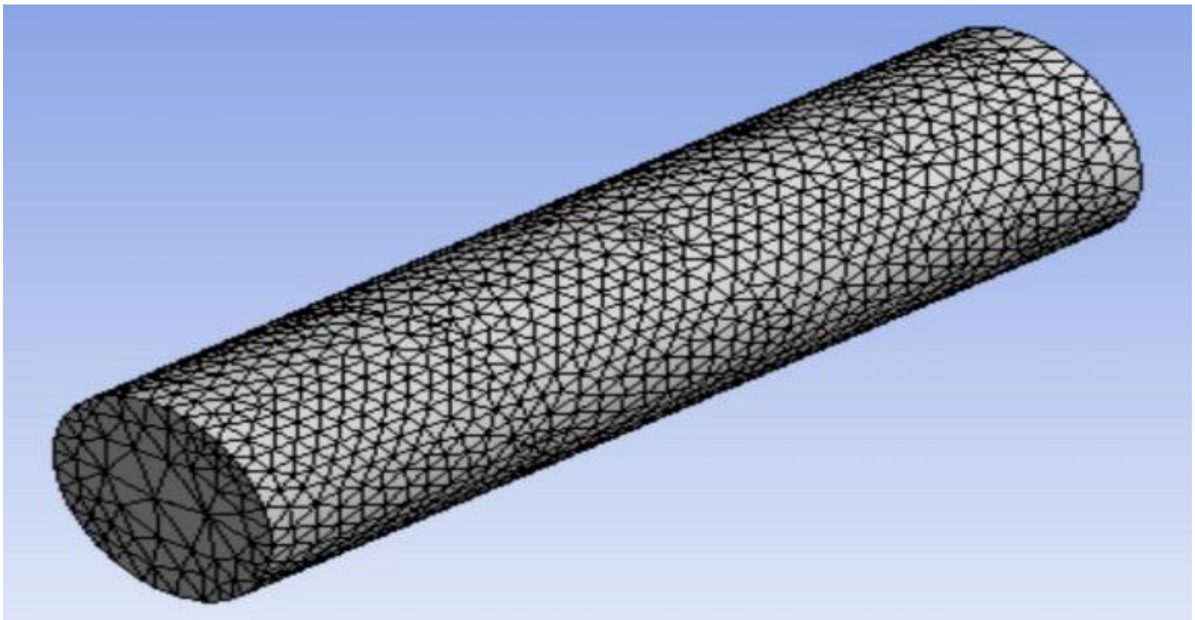


Figure 5.2: Meshing of cylinder in ANSYS

- **FLUENT SETUP:** Once mesh has been generated in Workbench meshing, it exported as a mesh file and imported into CFD code FLUENT. In present study 3 dimensional double precision fluent solvers with serial processing is used.

1. After reading mesh file into FLUENT, it is first scaled to proper unit if required. Then checked the quality of mesh which shows the orthogonal quality 0.44 as and aspect ratio as 11.2.

2. In this study pressure based transient solver is used with explicit formulation and gravitational field is enabled in this case.

3. In present study we are using air and gasoline as two different immiscible fluids, so model selected as multiphase with volume of fluid (VOF) formulation and the fluid flow considered as turbulent hence k-ε model is used.

4. Material used in present study is air and gasoline and aluminum is used for solid material.

5. In this study primary phase mentioned as air and secondary as gasoline. Cell zone condition considered as fluid.

6. Since the case in this study has no inlet or outlet boundaries, acceleration curve is imposed on the model in the form of a momentum source term. A user-defined code for momentum source. For linear motion Fig. 3.2 acceleration curve is imposed as UDF c-code.

7. In this study operating condition chosen as follows:
   - Operating pressure: 101325 Pa , Reference pressure location (0,1,4)
   - Gravitational acceleration : X=0 m/s^2,Y= -9.81 m/s^2, z=0 m/s^2
   - Operating density: 1.225 kg/m

8. Boundary condition given in simulation is as follows:
   - Baffle : wall
   - Baffle shadow : wall
   - Cylinder wall : wall

9. Solution Method is used in the present study is as follow:
   - Pressure-velocity coupling : Fractional step
   - Gradient : least square cell based
   - Pressure : Body force weighted
   - Momentum : Power law

- Volume fraction : Geo-Reconstruct
- Transient formulation :Non-iterative time advancement

10. Non-iterative relaxation factor :
    - Pressure: 0.8
    - Momentum: 0.6

11. Region of cell is adopted for filling the gasoline in tank by adaption method. Region is first adapted and then adapted cell is patched by gasolin
12. To display results in a 3D model, it will need surfaces on which the data can be displayed. FLUENT creates surfaces for all boundary zones automatically with iso-surface option. Iso-surface has been adopted for tracking the points on the free surface of water.

13. Time stepping method: Simulation of sloshing is explicit formulation and it is conditionally stable. Stability of above simulation is depending upon global courant number C. To avoid divergence global courant number should not exceed above 250. As we know that the global courant number depends on the mesh size, velocity field, and the time step size used for the transport equations. So to avoid `divergence in simulation i.e. (global courant number >250), provision of variable time stepping method is given. In variable time method

Global courant Number	2
Ending time (s)	20
Minimum time step size (s)	1e-07
Maximum time step size (s)	1e-03
Minimum step change factor	0.5
Maximum step change factor	1.5
Number of fixed time step	1
Time step (s)	1e-03

Table 5.1: Variable time stepping method

5.5 Closure:

This chapter discussed about computational fluid dynamics and its application in industry. In this chapter a comparison between experimental, analytical and CFD approach is discussed. A brief introduction to basic discretization scheme like FDM, FVM and FEA is given. This chapter also discussed about ANSYS-FLUENT solver theory and its various discretization scheme. A detailed procedure of geometry, meshing in ANSYS and FLUENT set-up is discussed.

Chapter 6

RESULTS AND DISCUSSION

RESULTS AND DISCUSSION

This chapter presents a detailed discussion about the result of the present study. In this study a computational simulation is carried out for a fuel tank when it is subjected to longitudinal and combined longitudinal and lateral acceleration field. Due to this accelerating motion, sloshing forces and moment developed in a tank which affects the stability of tank. So, in this study the effect of sloshing forces and moment is studied and the effect of these on stability is explained. Simulation is carried out as tank without baffle, with transverse baffle, and with transverse baffle with holes.

In this section graphs between longitudinal, vertical, lateral force Vs. Time and pitch and roll moment vs. time is shown with and without baffle.

6.1 Case I: When fuel tank is subjected to only longitudinal acceleration without baffle.

6.1.1 Fig.6.1 and Fig.6.2 shows the graph between longitudinal force vs. time for 40% fill and 80% fill condition and two different longitudinal acceleration i.e. a = 3.47 m/s^2 and a = 4.17 m/s^2.

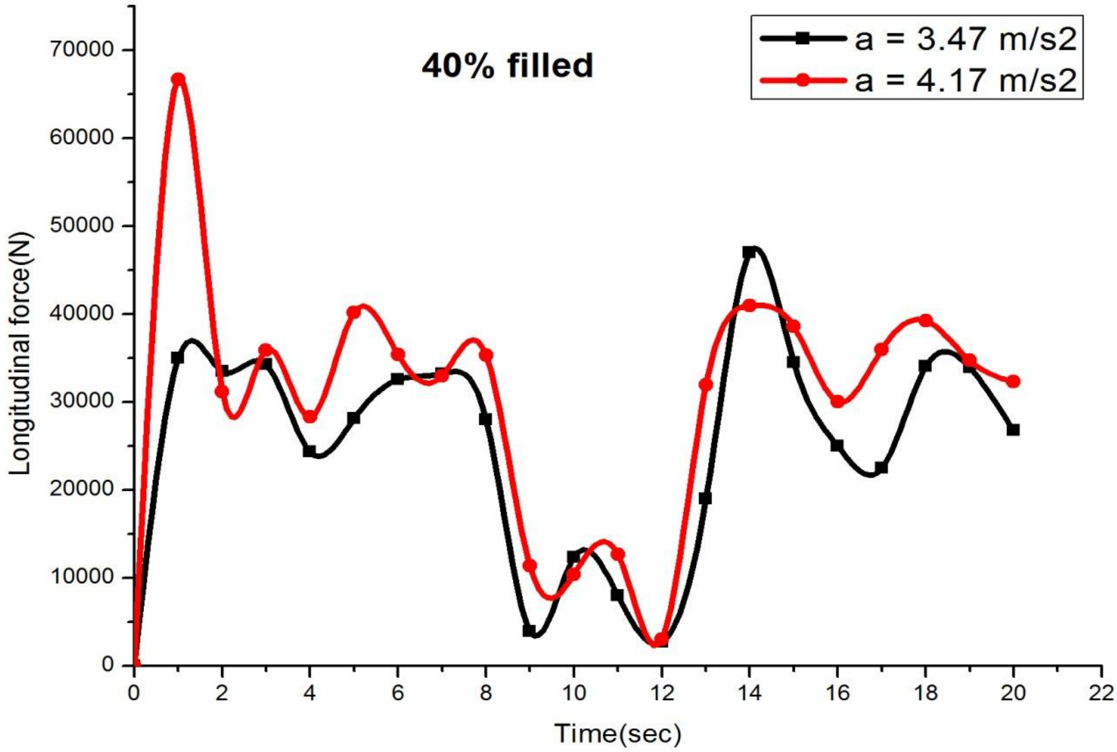


Fig. 6.1: longitudinal forces at 40% fill at 3.47 m/s^2 and at 4.17 m/s^2

From above Fig 6.1 it is clearly shows that variation of longitudinal forces is more when tank is subjected to value of higher acceleration as compared to the lower value of acceleration. The amplification of force increases at higher acceleration as compared the lower acceleration as time passes.

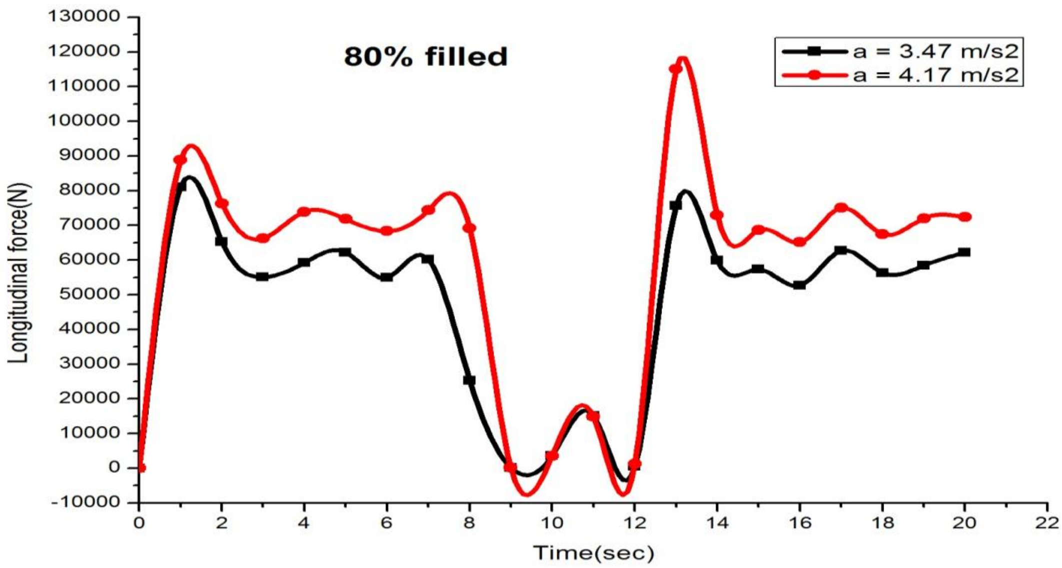


Fig. 6.2: longitudinal forces at 80% fill at 3.47 m/s^2 and at 4.17 m/s^2

The above Fig 6.2 shows the same result as explained for Fig 6.1. But in above graph it shows that amplification or variation of force is less as compared to force amplification when it is 40% fill, because of the large mass of fuel at 80% which does not slosh heavily .So from above two graphs it is concluded that the variation in longitudinal force is higher at higher acceleration and it will affect braking performance of vehicle severely.

6.1.2 Fig 6.3 and Fig.6.4 shows the graph between vertical force vs. time for 40% fill and

80% fill condition and two different longitudinal acceleration i.e. a = 3.47 m/s^2 and a = 4.17 m/s^2

The below two Fig.6.3 and Fig.6.4 it shows that variation in vertical force is very low or it is negligible in both cases of acceleration and fill level. This is because of tank is subjected to only longitudinal acceleration and this will not affect vertical force amplification.

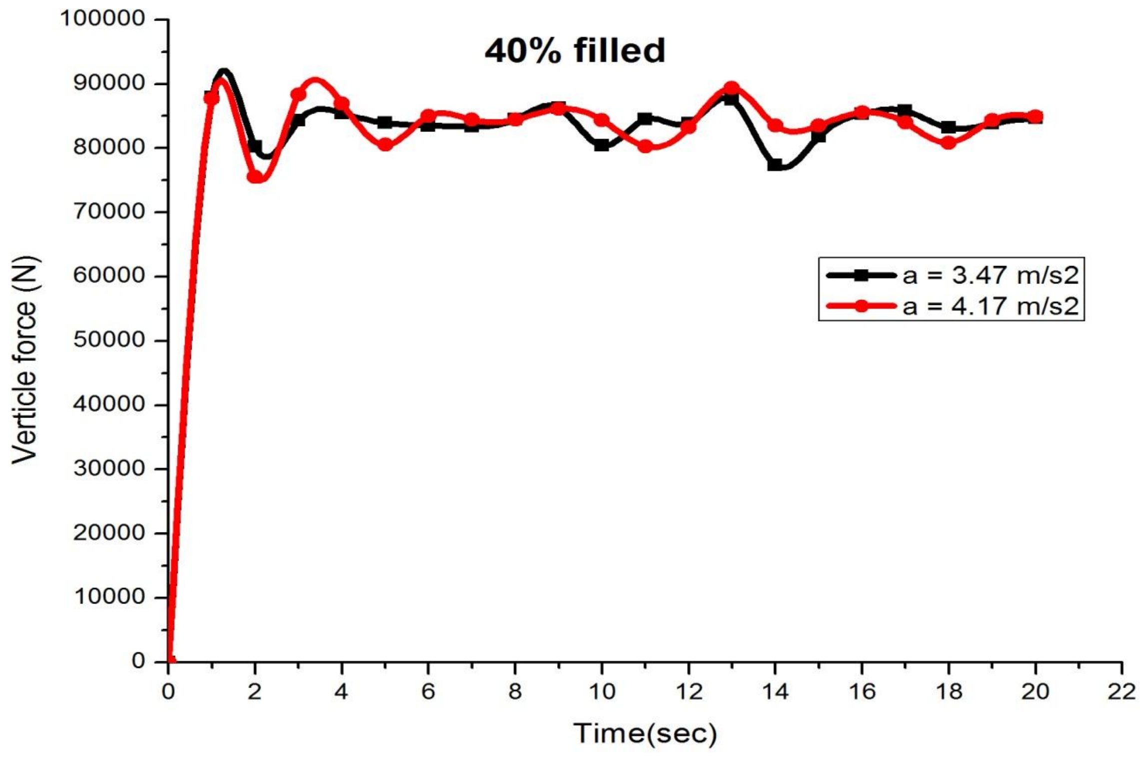


Fig. 6.3: Vertical forces at 40% fill at 3.47 m/s^2 and at 4.17 m/s^2

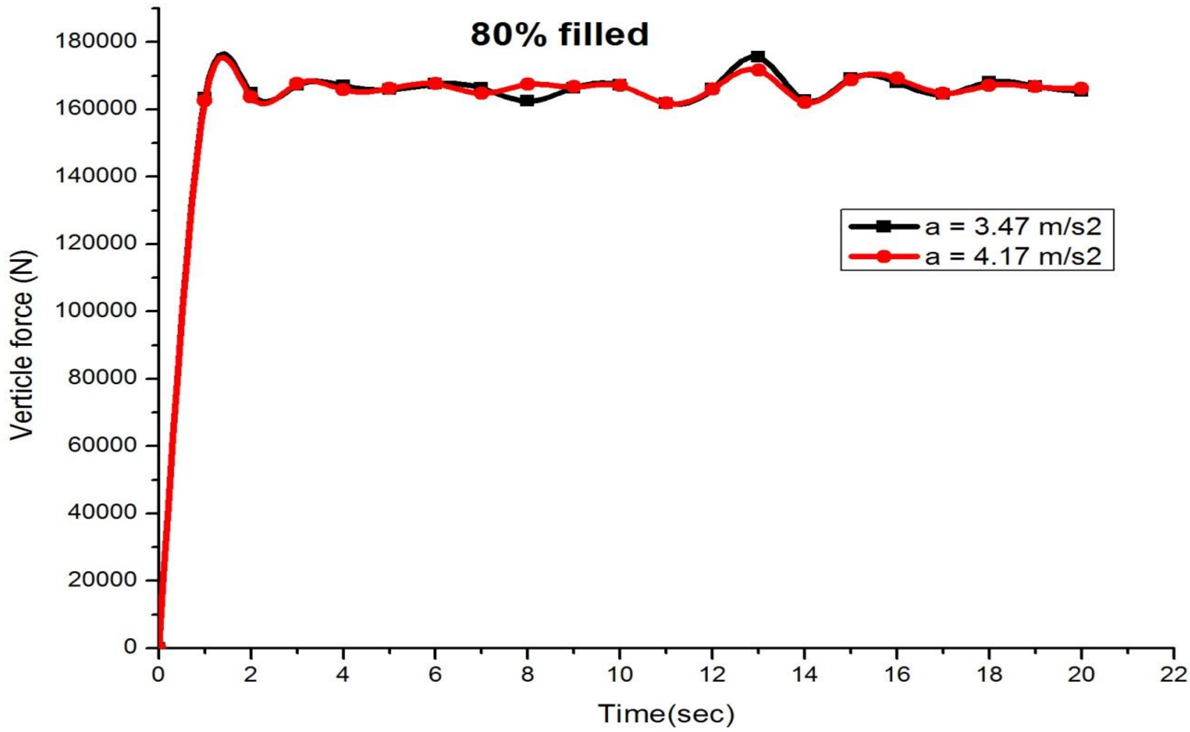


Fig. 6.4: vertical forces at 80% fill at 3.47 m/s^2 and at 4.17 m/s^2

6.1.3 Fig.6.5 and Fig.6.6 shows the plot between longitudinal force vs. time for a = 3.47 m/s^2 and a = 4.17 m/s^2 for 40% and 80% fill level.

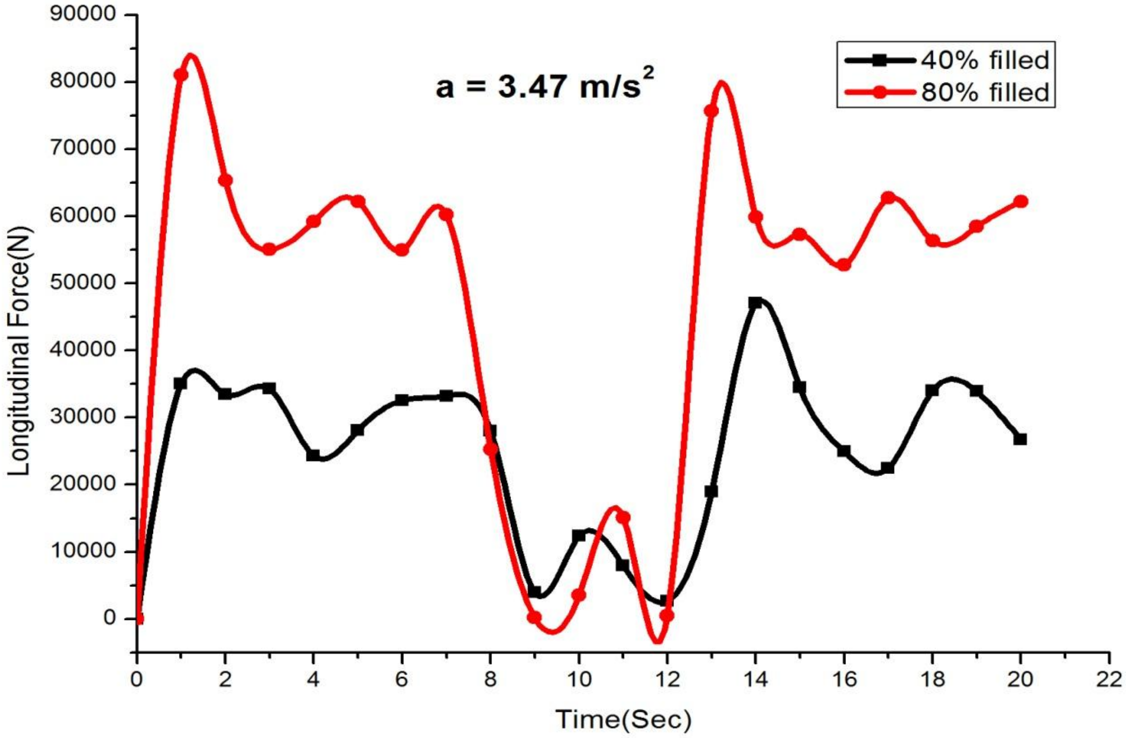
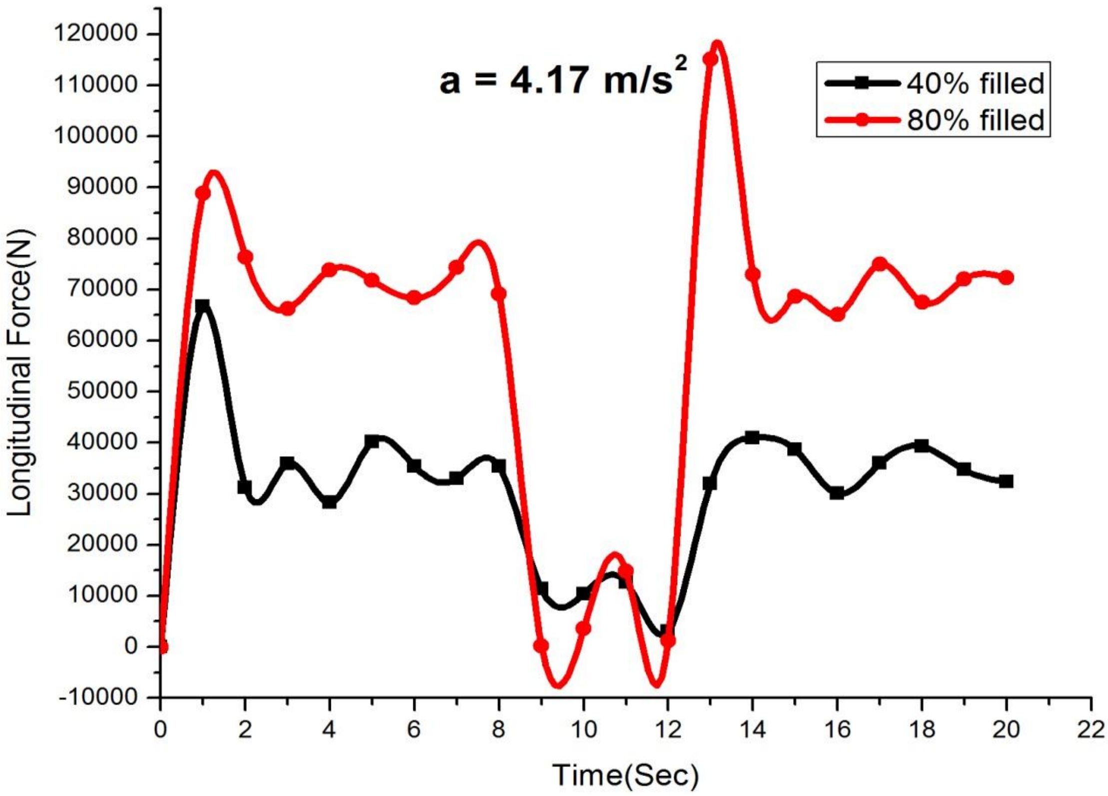


Fig. 6.5: Longitudinal forces at 3.47 m/s^2 fill at 40% and 80% fill



Fig, 6.6: Longitudinal forces at 4.17 m/s^2 fill at 40% and 80% fill

Fig 6.5 and Fig 6.6 it shows that variation of longitudinal forces is more at the lower fill level and higher acceleration. In this study it is at 40% fill and a= 4.17 m/s^2. So it is concluded that the decreasing fill level of fuel at higher acceleration affect the stability of the vehicle. In the present study it affects the braking performance of the vehicle when tank subjected to longitudinal acceleration alone. As the magnitude of lateral forces is small when tank subjected to longitudinal acceleration, hence in this case we didn't consider lateral force because it will not affect stability.

So to reduce the sloshing in a moving tank and to improve braking and steering performance, three lateral baffles are placed in the tank. The result of sloshing force and moment with baffle and without baffle is shown in below cases.

6.1.4 Fig.6.7 and Fig.6.8 shows the plot between longitudinal force vs. time for 40% fill level at a= 3.47m/s^2 and a= 4.17 m/s^2 with and without baffle.

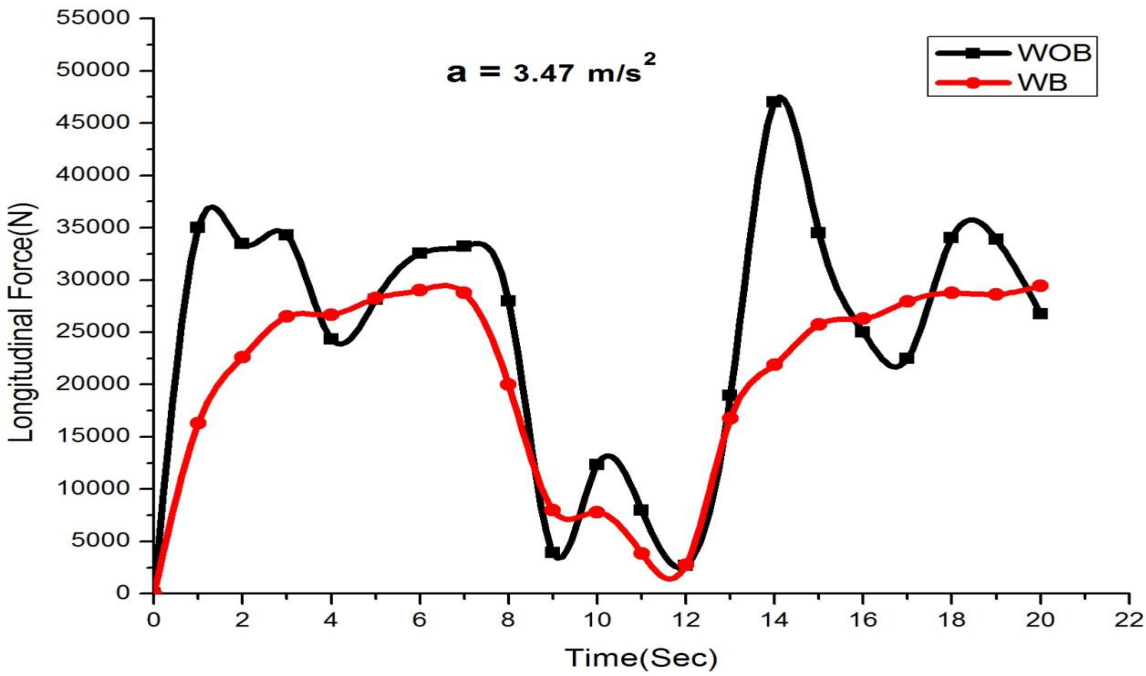


Fig. 6.7: Longitudinal Force at 3.47 m/s^2 with and without baffle at 40% fill

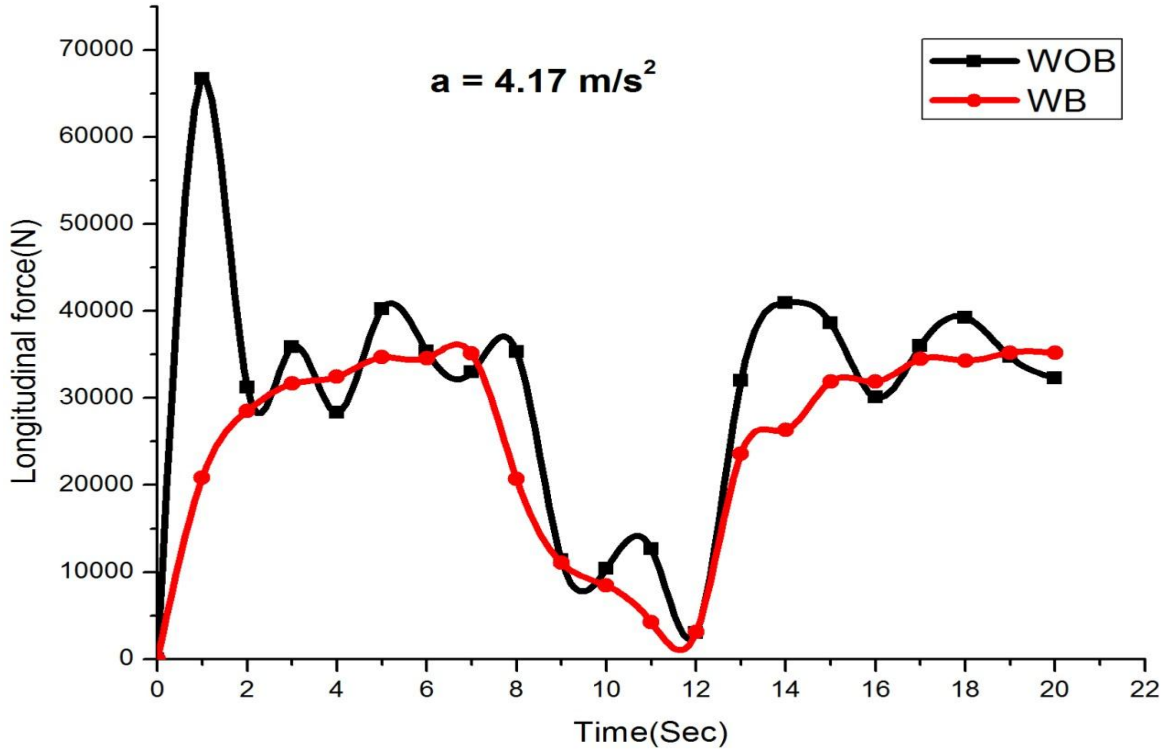


Fig. 6.8: Longitudinal Force at 4.17 m/s^2 with and without baffle at 40% fill

Fig 6.7 and Fig 6.8 shows that by placing three transverse baffles there is a reduction in variation and magnitude of longitudinal forces. It is because of fluid separated around baffle which results into dissipation of energy which reduces force variation. The

above figure also shows that by placing baffle the reduction in force variation is more effective at higher acceleration.

6.1.5 Fig.6.9 and Fig.6.10 shows the plot between longitudinal force vs. time for 80% fill level at a = 3.47 m/s^2 and a = 4.17 m/s^2 with and without baffle.

Fig 6.9 and Fig 6.10 also shows the same result which as shown in above case. In this case force variation reduces by placing the baffle. But the reduction in forces by placing baffle is more at lower fill and high acceleration. So placing three transverse baffles it improves breaking performance as well.

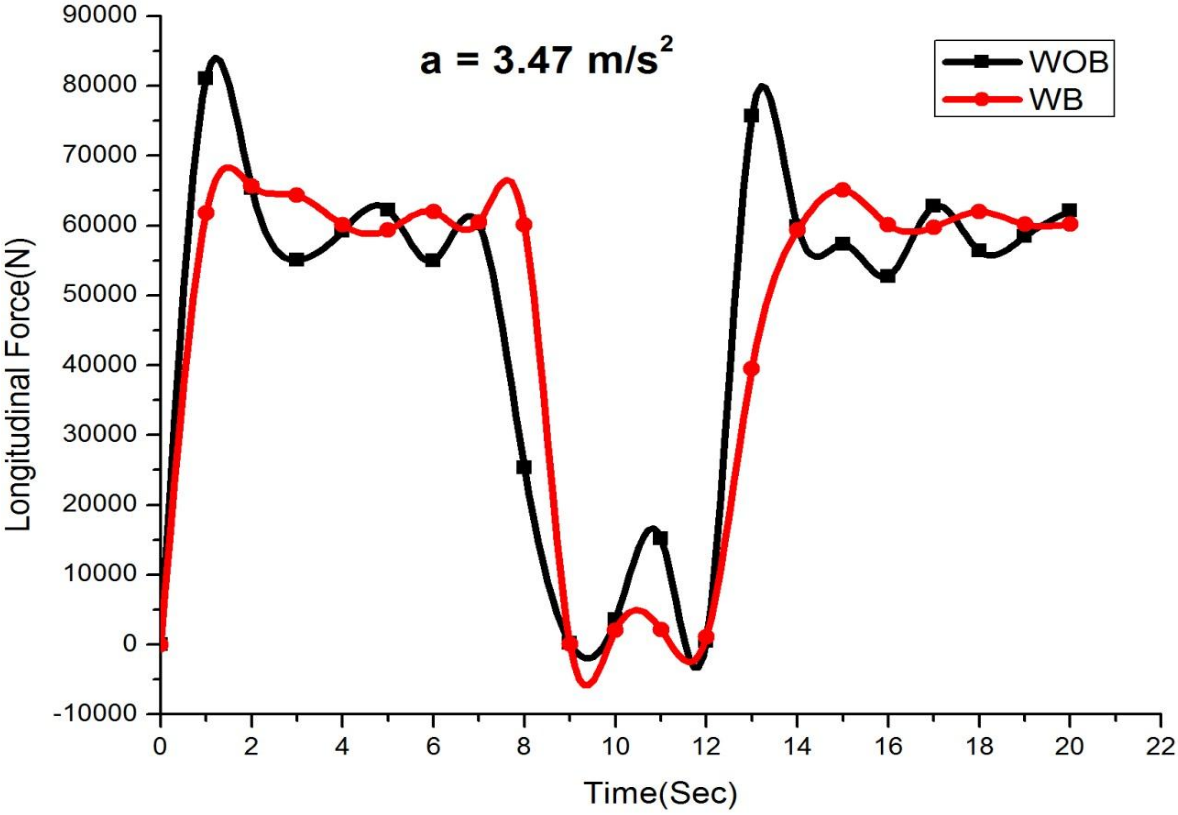


Fig. 6.9: Longitudinal Force at 3.47 m/s^2 with and without baffle at 80% fill

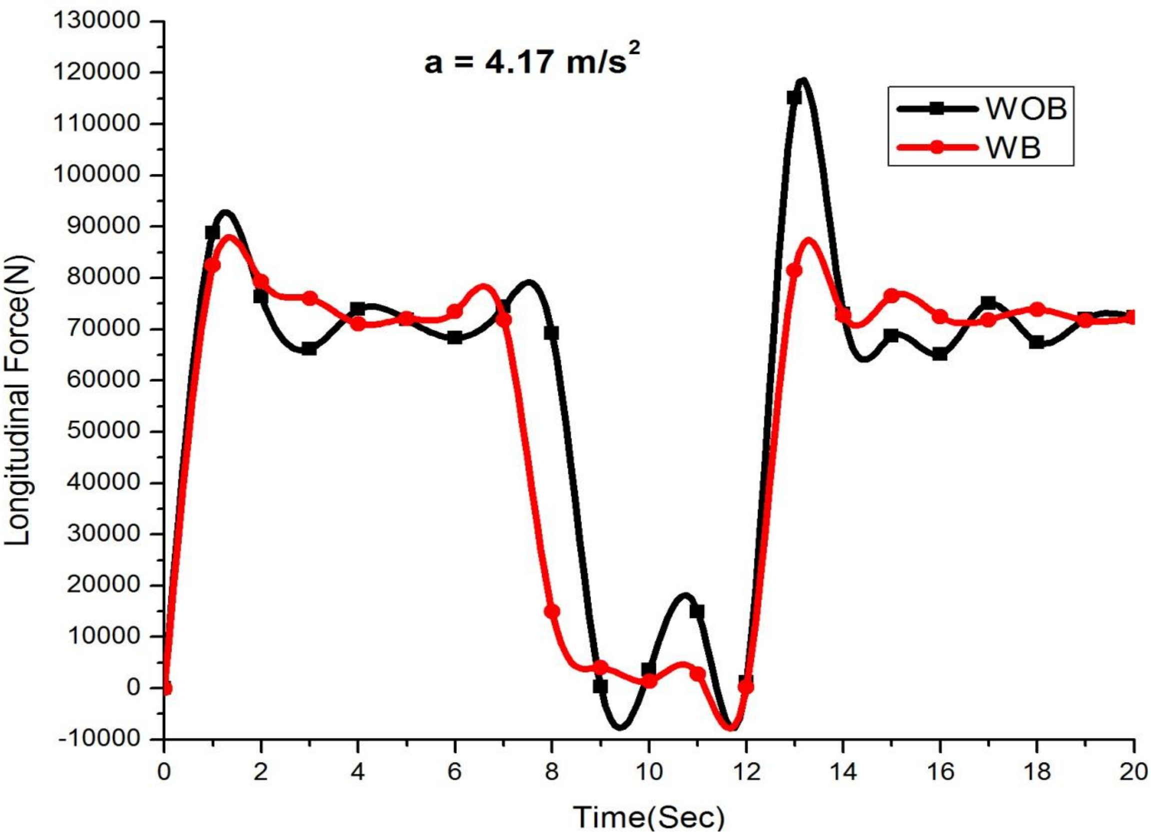


Fig. 6.10: Longitudinal Force at 4.17 m/s^2 with and without baffle at 80% fill

6.1.6 Fig.6.11 and Fig.6.12 shows the plot between vertical force vs. time for 40% fill, at a= 3.47 m/s^2 and for 40% fill, at a= 4.17 m/s^2 with and without baffle.

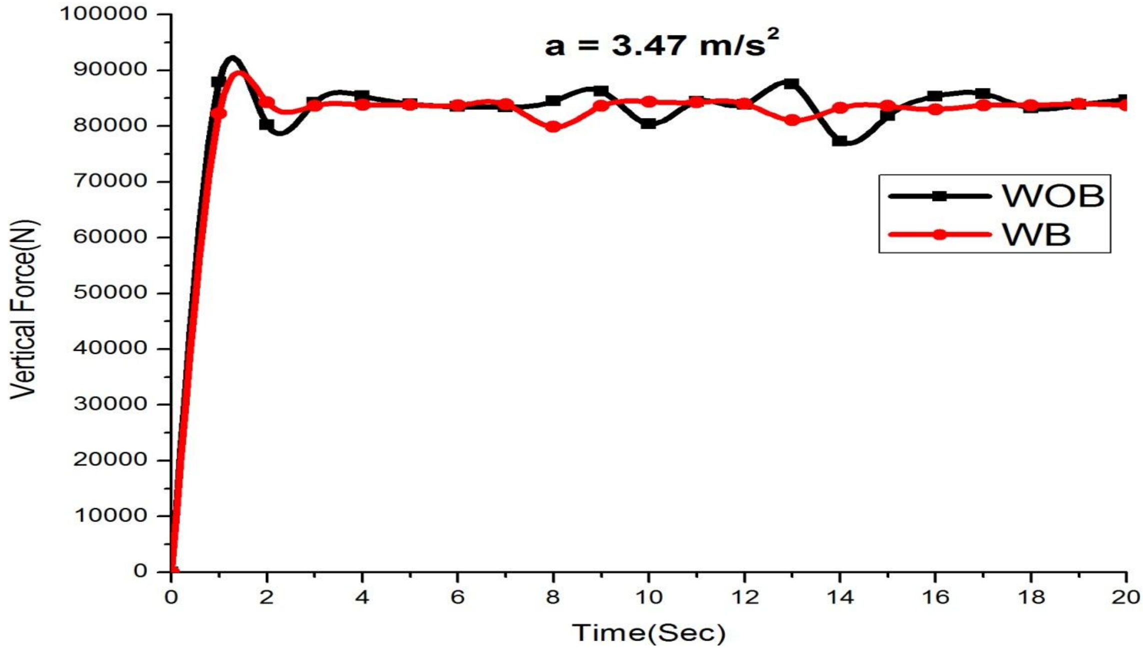


Fig. 6.11: Vertical Force at 3.47 m/s^2 with and without baffle at 40% fill

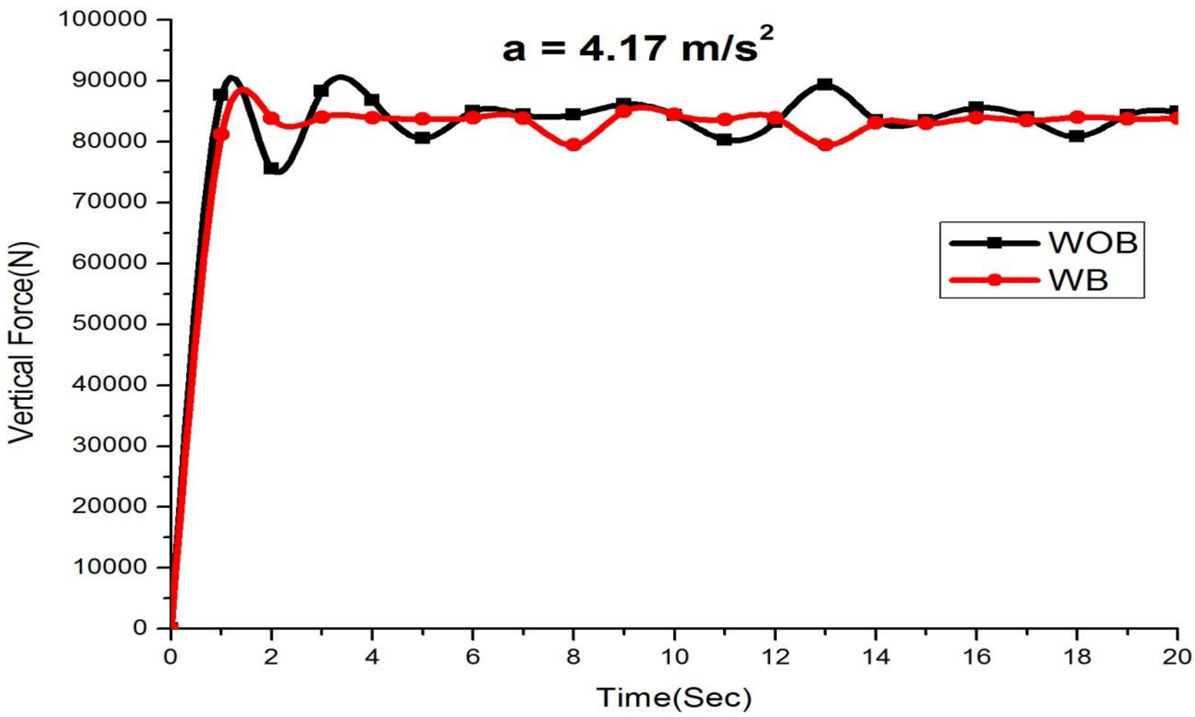


Fig. 6.12: Vertical Force at 4.17 m/s^2 with and without baffle at 40% fill

The above graph shows that there is not much (negligible) reduction in vertical force variation by placing baffle.

6.1.7 Fig.6.13 and Fig.6.14 shows the plot between pitch moment vs. time for 40% fill level at a = 3.47 m/s^2 and a = 4.17 m/s^2 with and without baffle.

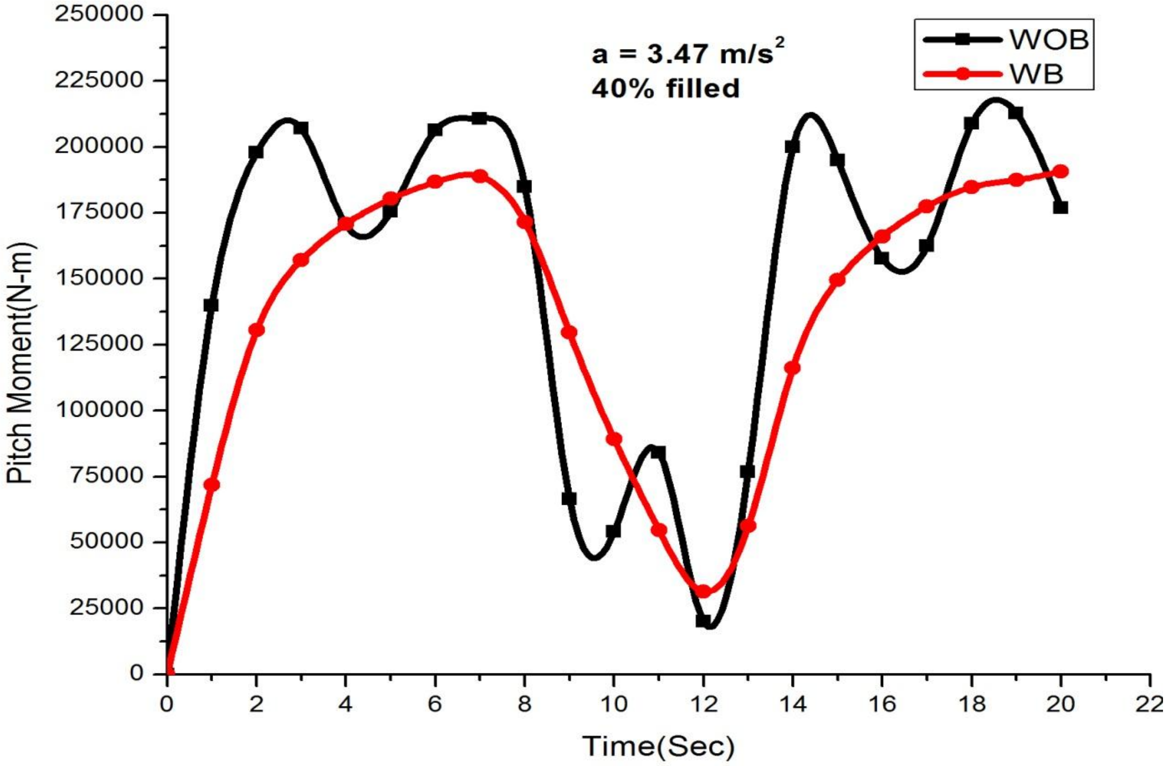


Fig. 6.13: Pitch moments at 3.47 m/s^2 with and without baffle at 40% fill

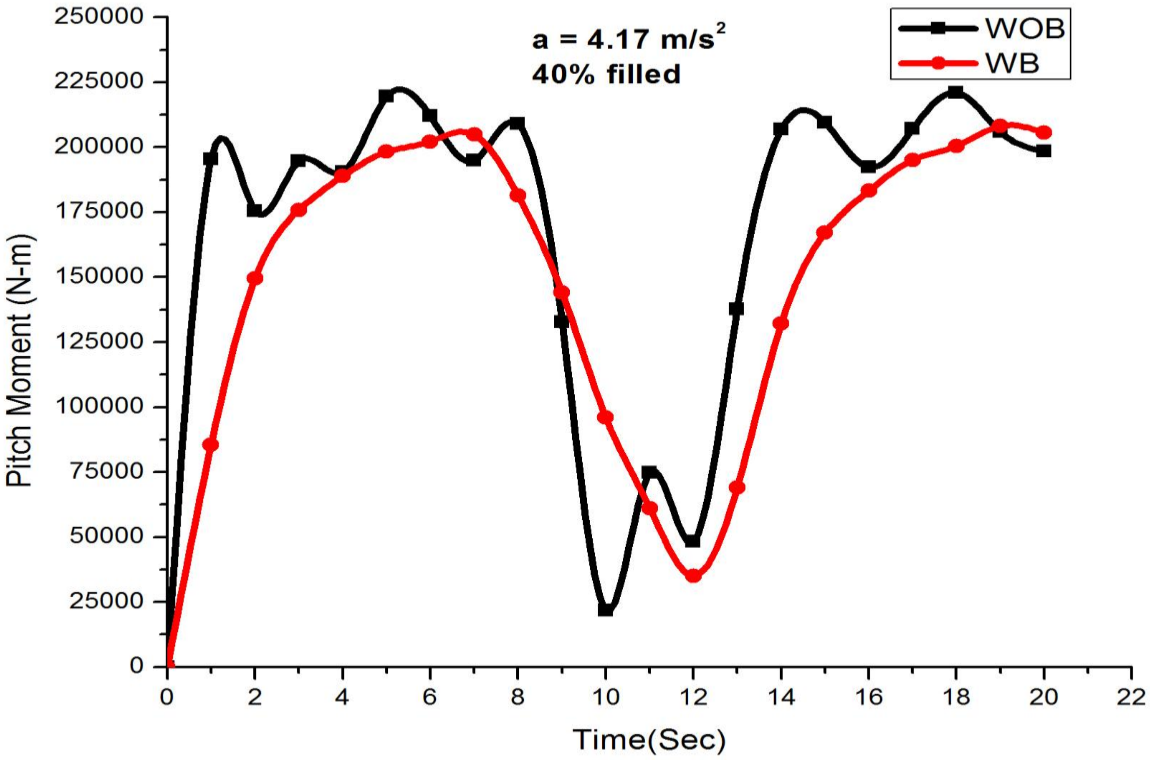


Fig. 6.14: Pitch Moment at 4.17 m/s^2 with and without baffle at 40% fill

6.1.8 Fig.6.15 and Fig.6.16 shows the plot between pitch moment vs. time for 80% fill level at a = 3.47 m/s^2 and a = 4.17 m/s^2 with and without baffle.

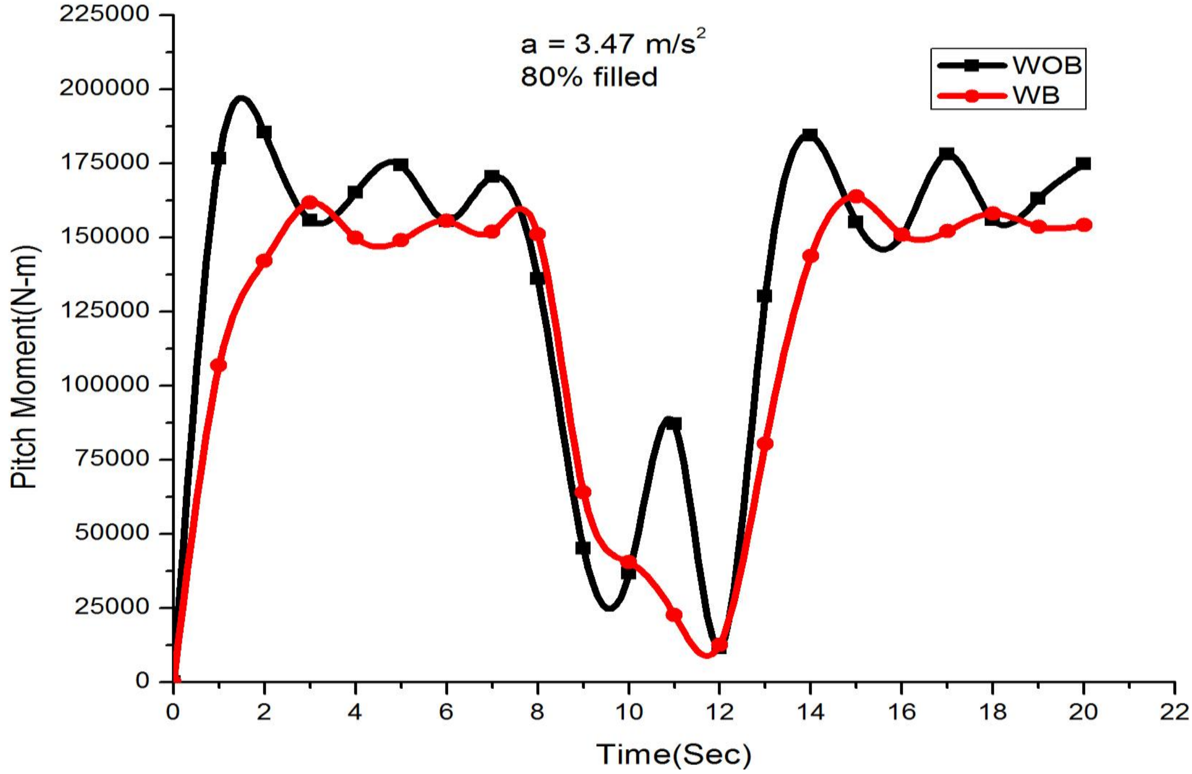


Fig 6.15: Pitch Moment at 3.47 m/s^2 with and without baffle at 80% fill

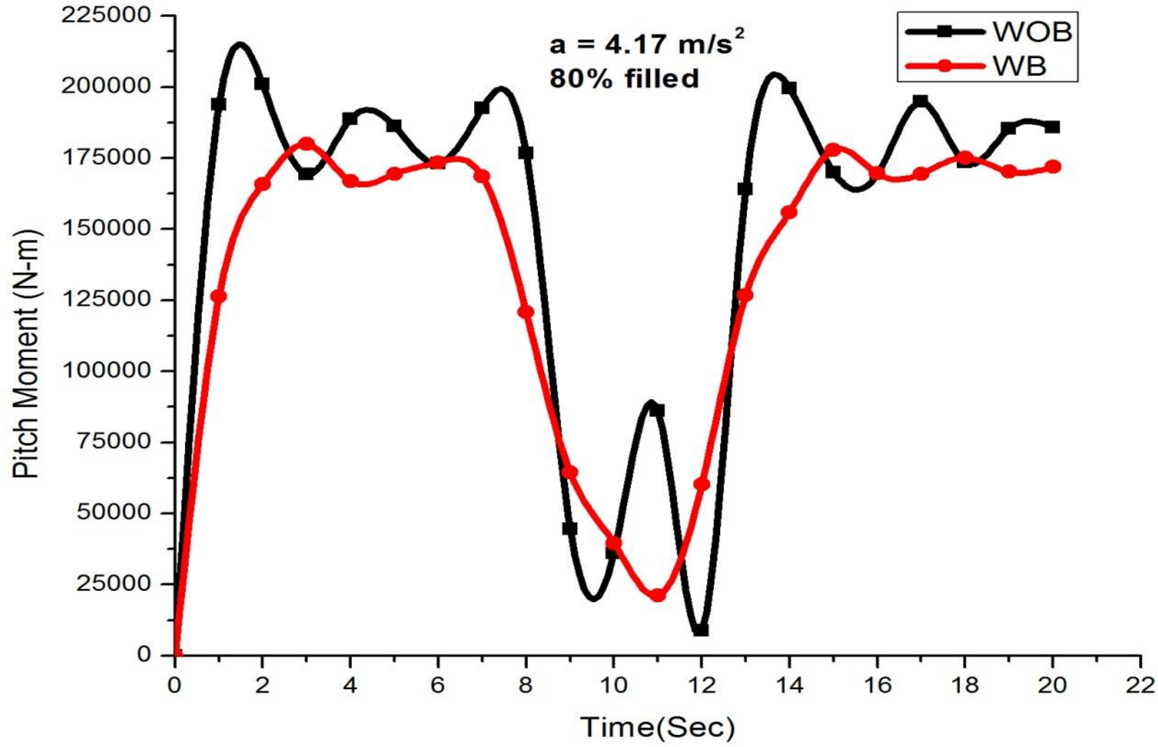


Fig 6.16: Pitch Moment at 4.17 m/s^2 with and without baffle at 80% fill

The all above four graphs shows that variation in pitch moment is reduced by placing three transverse baffles. This pitch moment affects the pitch plane stability of the vehicle. The above graph also shows that variation in pitch moment is higher at the lower fill level and reduction in pitch moment is higher at lower fill level. From case-I it is concluded that variation in the forces and moment is higher at the lower fill level and high acceleration without baffle. By introducing three transverse baffle slosh forces and moment reduces significantly at the lower fill level and high acceleration.

6.2 Case II: When the fuel tank is subjected to combined longitudinal and lateral acceleration with transverse baffle and without baffle.

6.2.1 Fig.6.17 and Fig.6.18 shows the plot between longitudinal force vs. time for 40% fill level a_z = 3.47 m/s^2 and a_x = 1.5m/s^2 and for 80% fill level at a_z = 3.47 m/s^2 and a_x= 1.5 m/s^2.

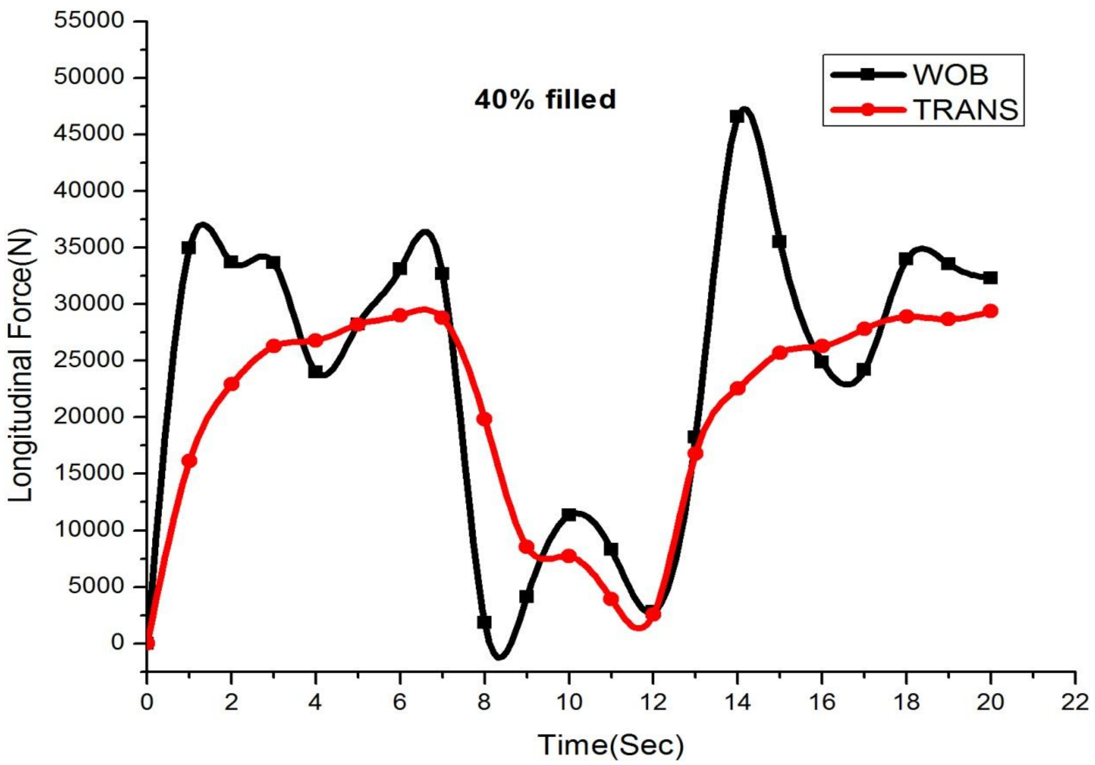


Fig. 6.17: longitudinal forces for 40% fill level at a_z= 3.47 m/s^2 and a_x= 1.5m/s^2

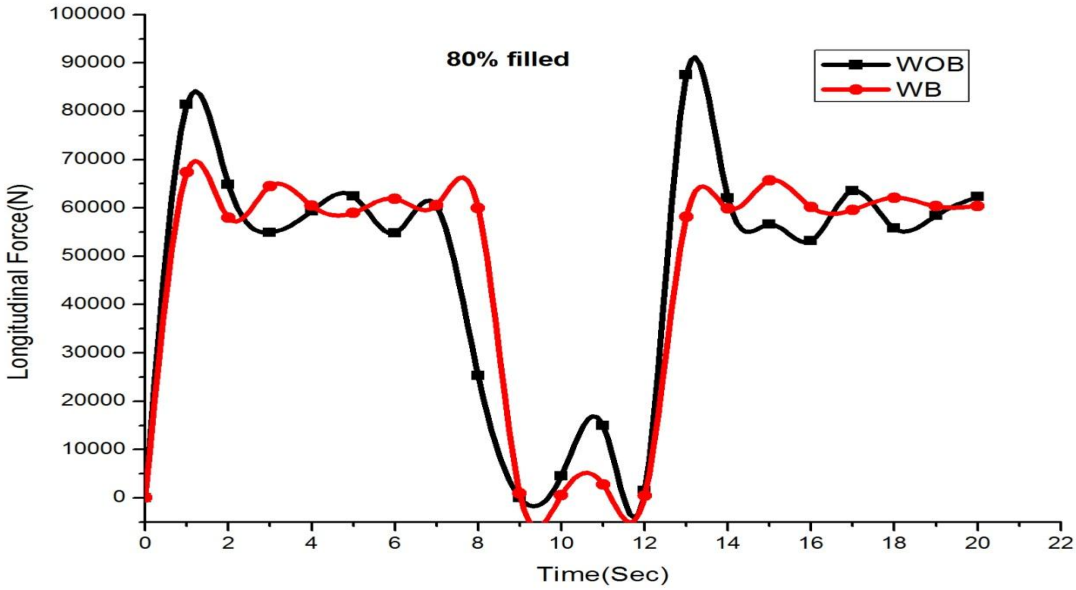


Fig. 6.18: longitudinal forces for 80% fill level at a_z= 3.47 m/s^2 and a_x= 1.5m/s^2

The above Fig.6.17 shows that variation in longitudinal force when subjected to combined longitudinal and lateral acceleration is reduced by placing three transverse baffle. While comparing with Fig 6.18, variation is more at the lower fill level. Most of results same as in case of only longitudinal acceleration i.e. variation is higher at the lower fill level and higher acceleration.

6.2.2 Fig.6.19 and Fig.6.20 shows the plot between lateral force vs. Time for 40% fill level at a_z = 3.47 m/s^2 and a_x = 1.5 m/s^2 and for 80% fill level at a_z = 3.47 m/s^2 and a_x = 1.5 m/s^2.

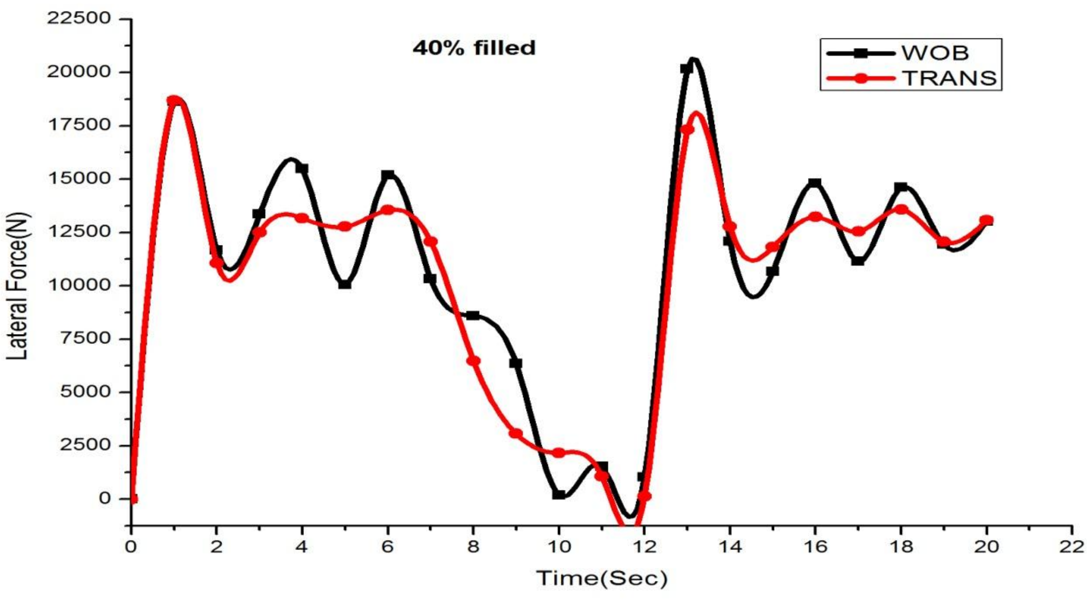


Fig.6.19: Lateral force for 40% fill level at a_z = 3.47 m/s^2 and a_x= 1.5 m/s^2

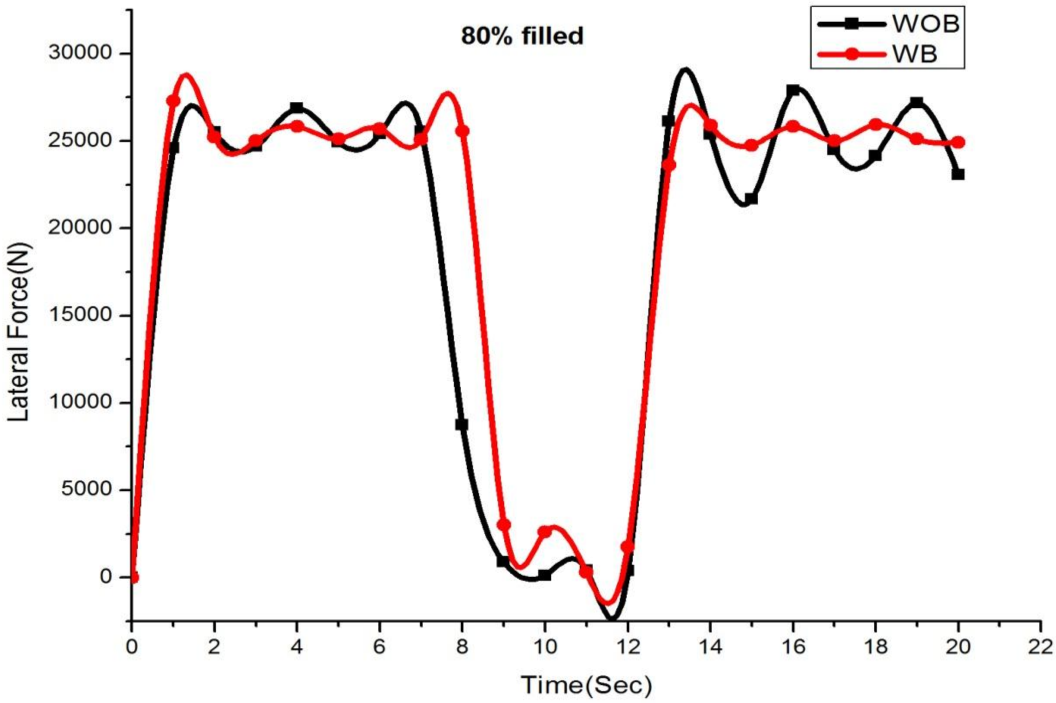


Fig.6.20: Lateral force for 80% fill level at $a_z = 3.47\ m/s^2$ and $a_x = 1.5\ m/s^2$

From above two figures it is shown that variation of lateral force when tank subjected to combined lateral and longitudinal acceleration is not reduced considerably by placing transverse baffle. This is because of transverse baffle providing less resistance to lateral motion of fuel. These lateral forces affect the rolling motion of vehicle and it also reduces steering performance. So from case-II it is concluded that longitudinal force and pitch moment reduced by the great amount while lateral force variation is not reduced by placing three transverse baffles.

6.3 Case III: Comparison between transverse, transverse baffle with holes when, Fuel tank is subjected to combined longitudinal and lateral acceleration.

6.3.1 Fig.6.21 and Fig.6.22 shows the graph between longitudinal force vs. time for 40% fill level at $a_z = 3.47\ m/s^2$ and $a_x = 1.5 m/s^2$ and for 80% fill level at $a_z = 3.47\ m/s^2$ and $a_x = 1.5\ m/s^2$.

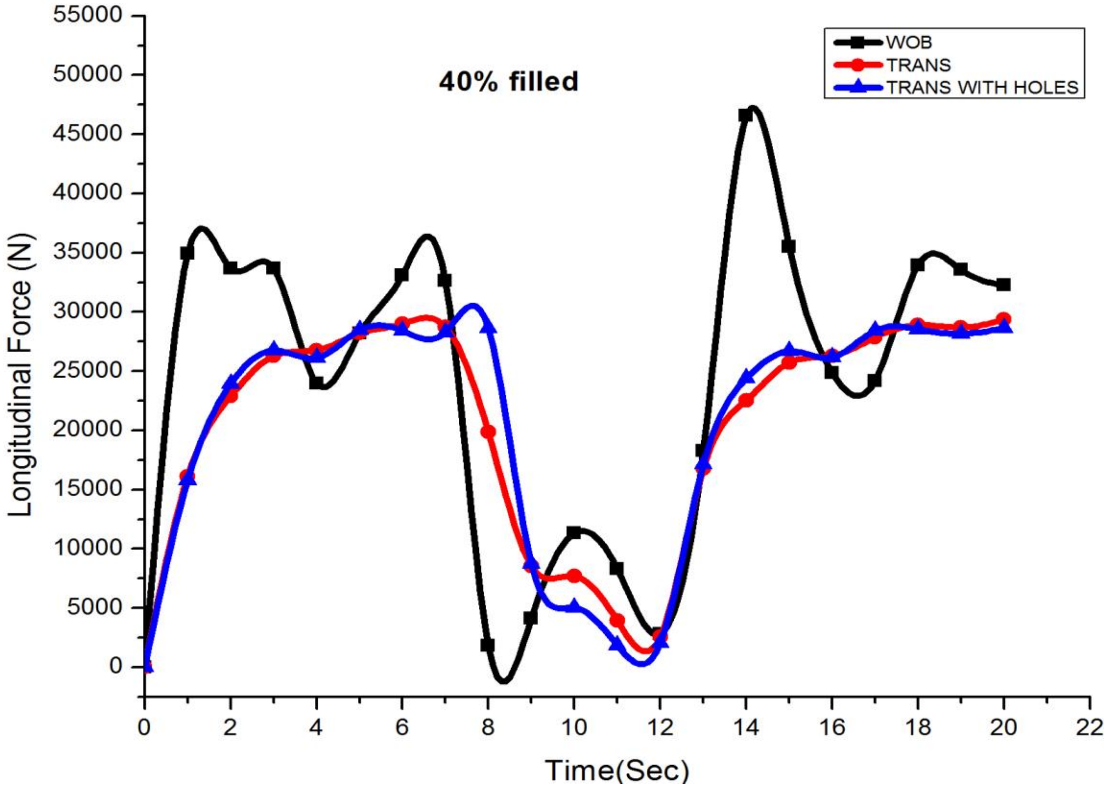


Fig 6.21: longitudinal forces for 40% fill level at a_z= 3.47 m/s^2 and a_x= 1.5m/s^2

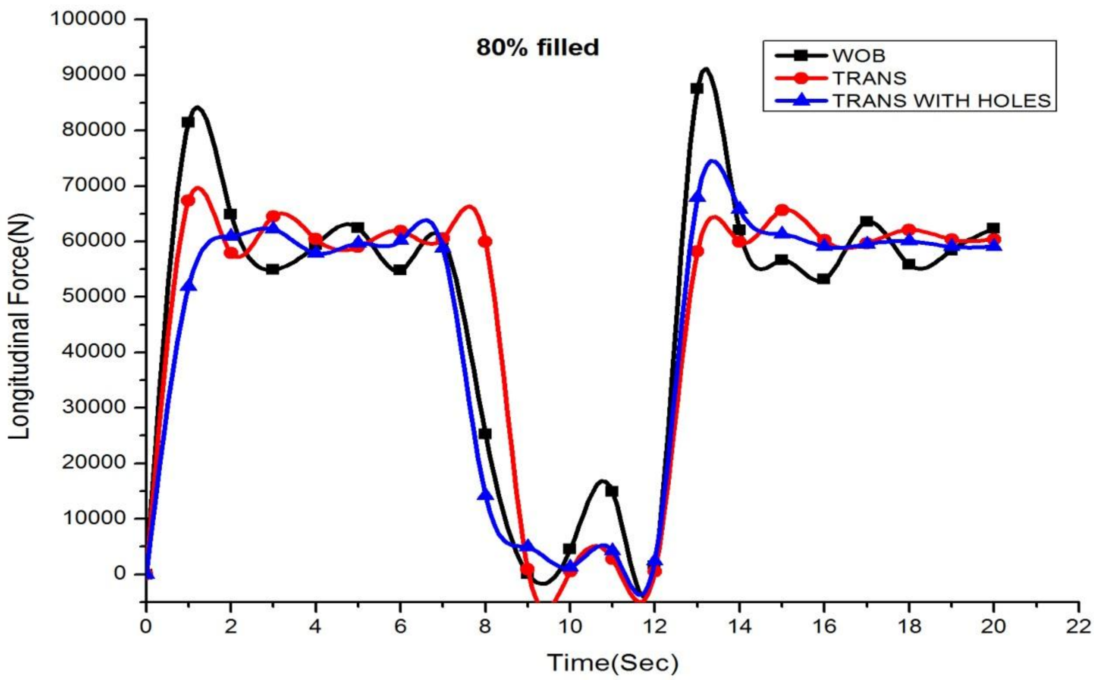


Fig. 6.22: longitudinal forces for 80% fill level at a_z= 3.47 m/s^2 and a_x= 1.5m/s^2

From above graph it is shown that longitudinal force variation by placing transverse baffle with holes is not affected too much as compared to transverse baffle.

6.3.2 The Fig.6.23 and Fig.6.24 shows the graph between lateral forces vs. time for 40% fill level at a_z = 3.47 m/s^2 and a_x = 1.5 m/s^2 and for 80% fill level at a_z = 3.47 m/s^2 and a_x = 1.5 m/s^2

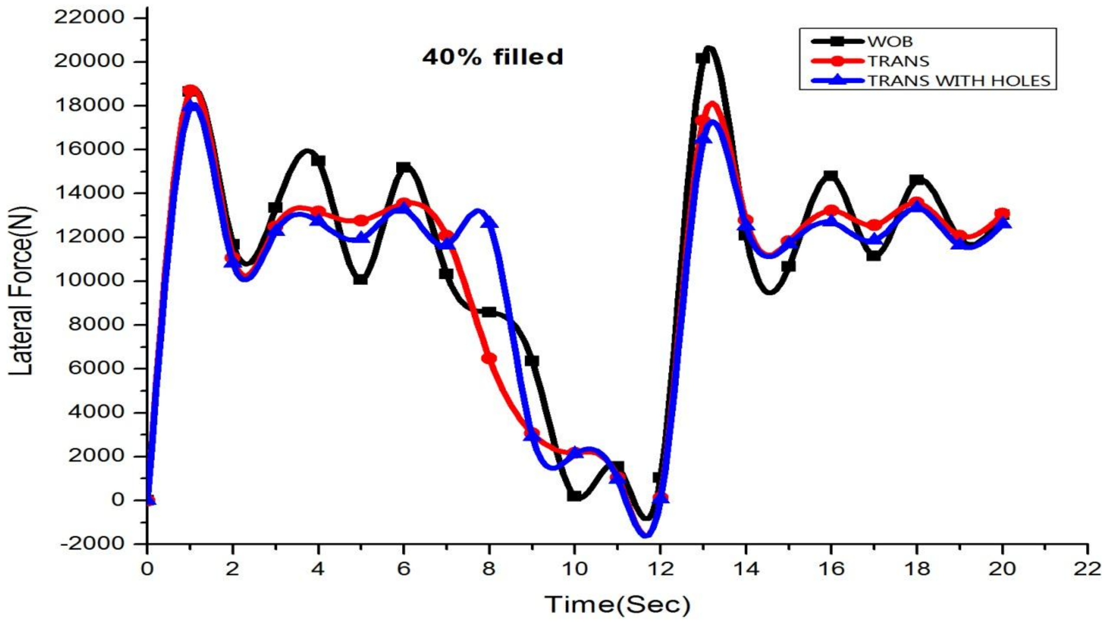


Fig.6.23: Lateral force for 40% fill level at a_z = 3.47 m/s^2 and a_x= 1.5 m/s^2

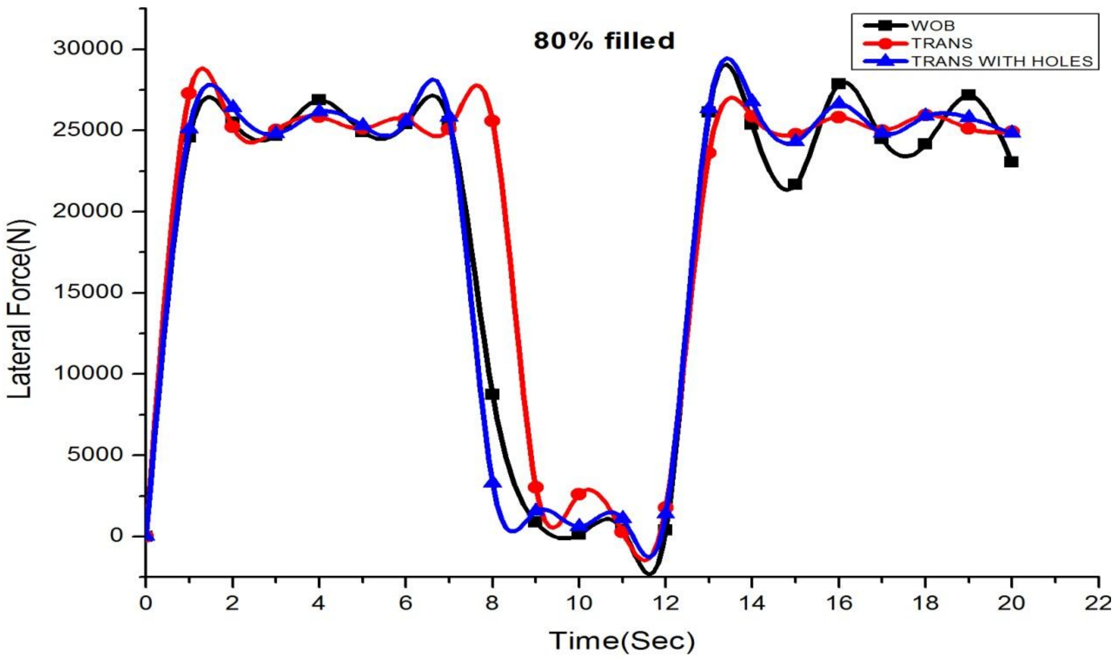


Fig.6.24: Lateral force for 80% fill level at a_z = 3.47 m/s^2 and a_x= 1.5 m/s^2

Fig.6.23 and Fig. 6.24 shows the time histories of lateral force response at 40% and 80% fill level. The result shows that transverse baffle and transverse baffle with holes is not effective in reducing lateral sloshing forces.

6.3.3 The Fig.6.25 and Fig.6.26 shows the graph between pitch moment vs. time for 40% fill level at $a_z = 3.47\ m/s^2$ and $a_x = 1.5\ m/s^2$ and for 80% fill level at $a_z = 3.47\ m/s^2$ and $a_x = 1.5 m/s^2$.

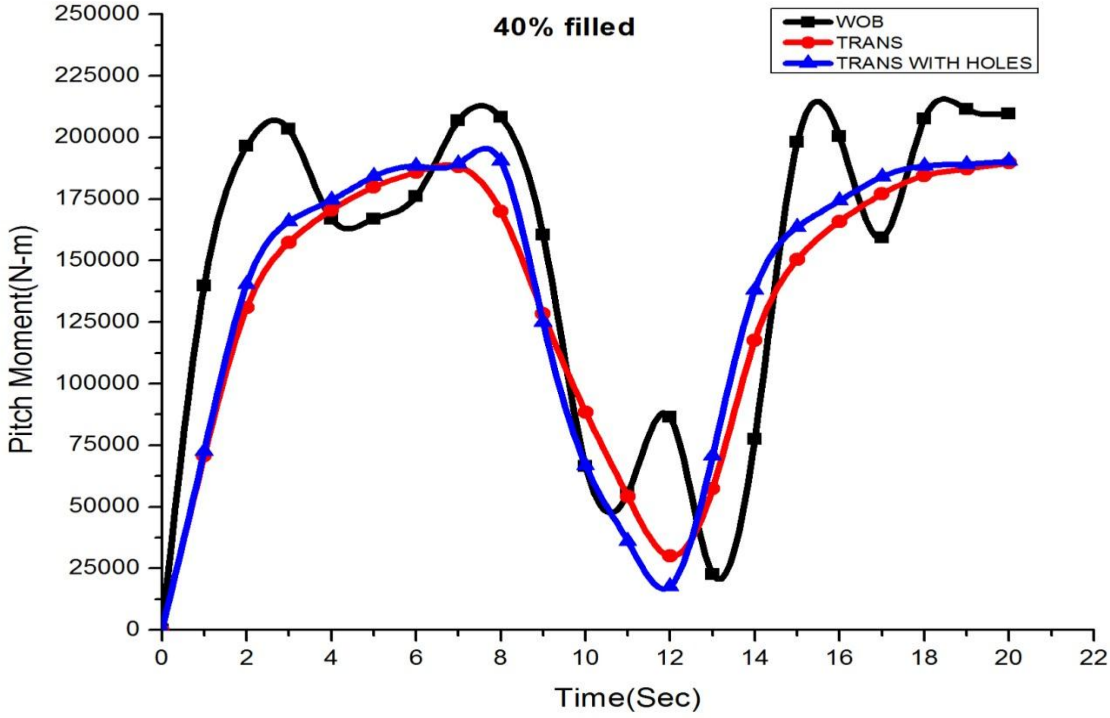


Fig.6.25: Pitch Moment for 40% fill level at $a_z = 3.47\ m/s^2$ and $a_x = 1.5\ m/s^2$

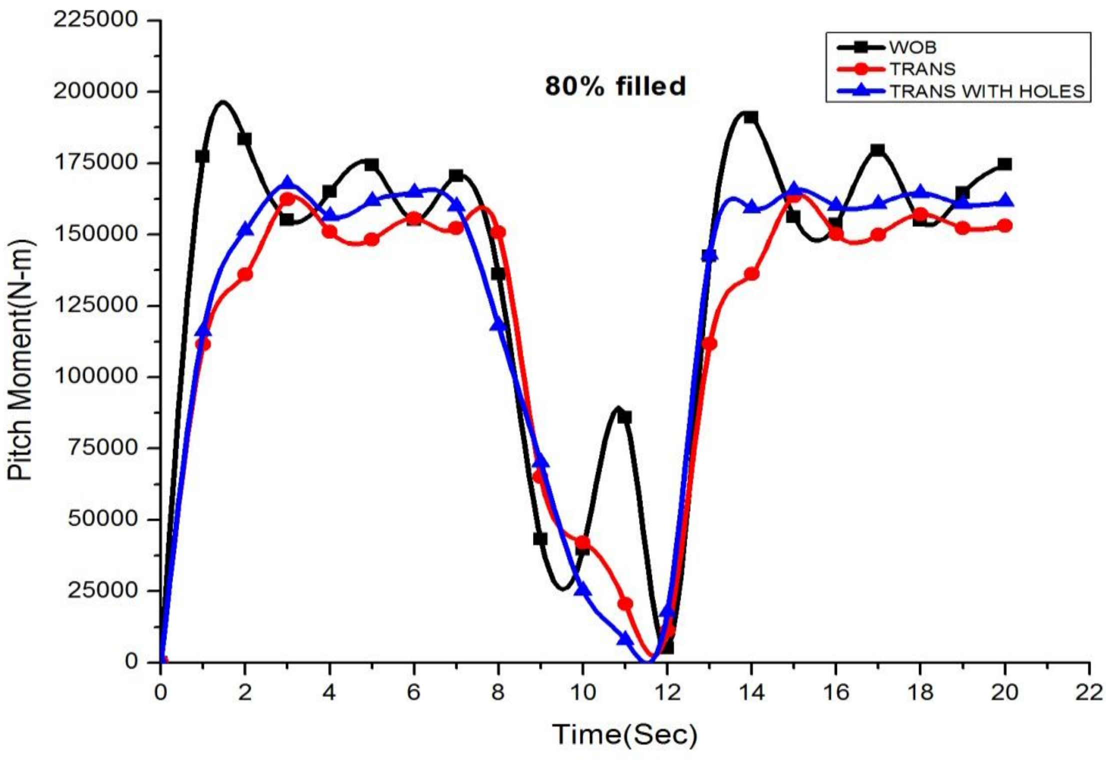


Fig.6.26: Pitch Moment for 80% fill level at $a_z = 3.47\ m/s^2$ and $a_x = 1.5\ m/s^2$

6.3.4 Fig.6.27 and Fig.6.28 shows the graph between Roll moment vs. time for 40% fill level at a_z = 3.47 m/s^2 and a_x = 1.5 m/s^2 and for 80% fill level at a_z = 3.47 m/s^2 and a_x = 1.5m/s^2.

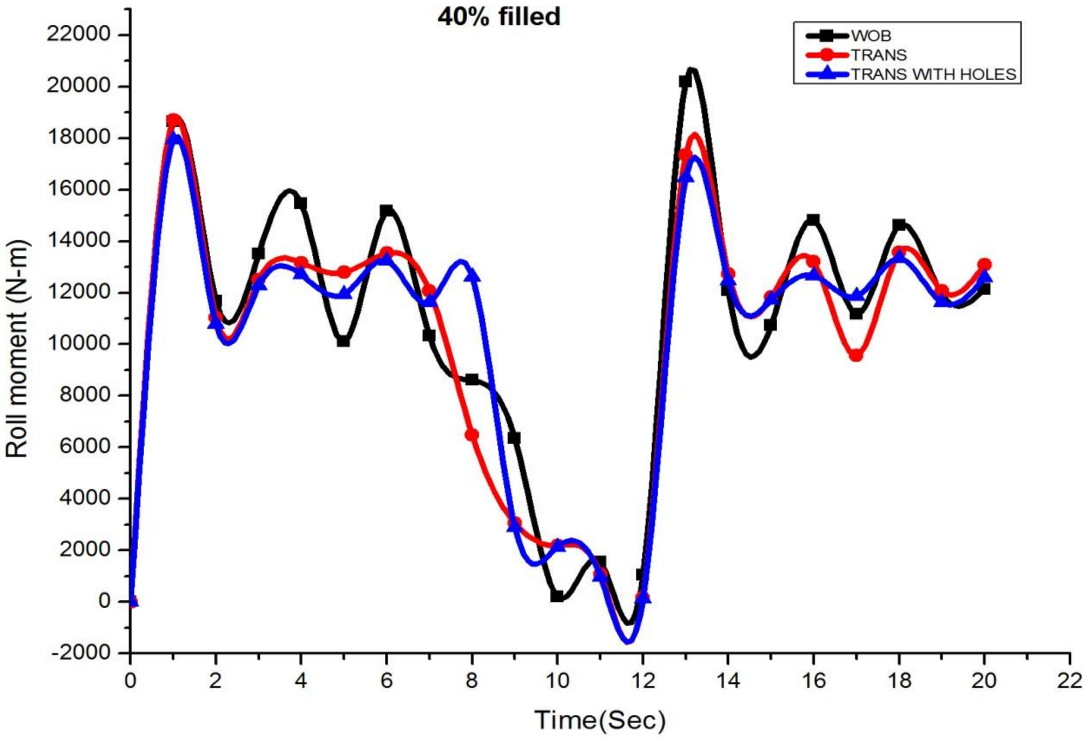


Fig.6.27: Roll Moment for 40% fill level at a_z = 3.47 m/s^2 and a_x= 1.5 m/s^2

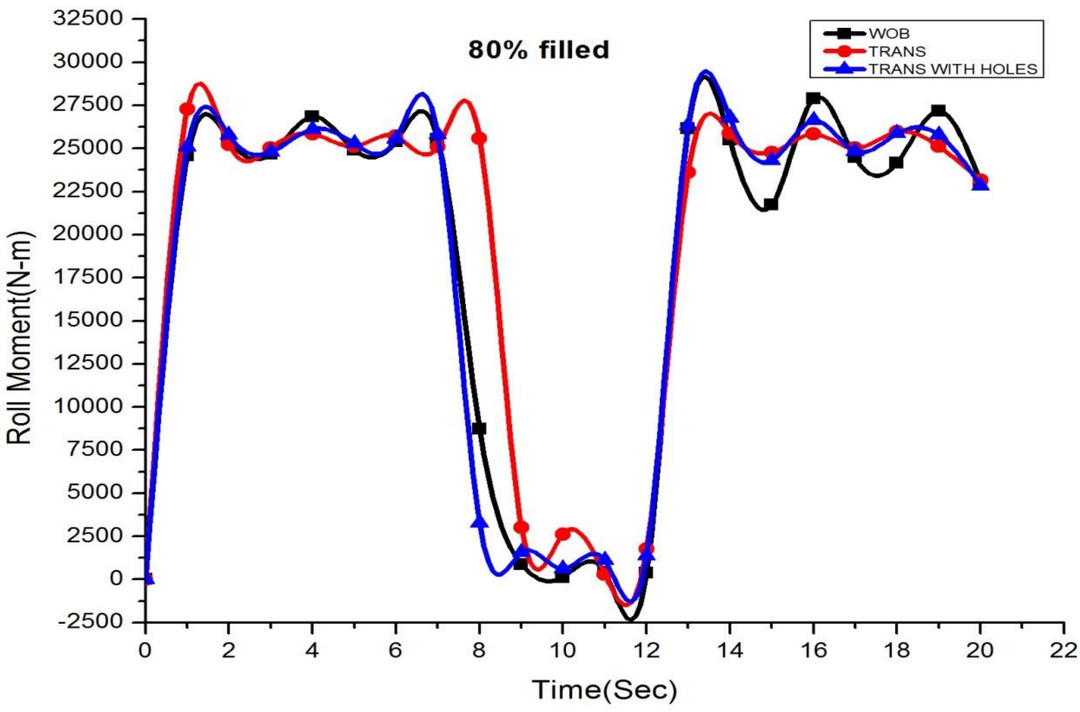


Fig.6.28: Roll Moment for 80% fill level at a_z = 3.47 m/s^2 and a_x= 1.5 m/s^2

Fig.6.25 and 6.26 shows the time histories of pitch moment response at 40% and 80% fill level under simultaneous longitudinal and lateral/ longitudinal acceleration. The result shows that the fluctuation of peak moment diminishes more rapidly in transverse baffle with holes than transverse baffle because fuel is being passed in holes very smoothly. Hence slosh load shift in pitch plane is low in case of the transverse baffle with holes which improve braking performance of the vehicle.

Fig.6.27 and 6.28 shows the time histories of roll moment response at 40% and 80% fill level under simultaneous longitudinal and lateral/ longitudinal acceleration. The result shows that both the baffles have no effects on roll moment because they provide very less resistance to roll moment due their orientation and placement.

6.4 Closure:

In this section result for various cases are discussed with graphs. The result shows that transverse baffle with holes is most effective in pitch plane for reducing slosh forces and moment.

Chapter 7

Conclusions and Future Work

Conclusions and Future Work

This chapter includes the conclusion, which I have made after this investigation and scope for future work.

7.1 Conclusion

The sloshing force and moment caused by the fuel slosh in the partially filled cylindrical fuel tank subjected to lateral, longitudinal and combined lateral/longitudinal accelerations are investigated .

The simulation of the present problem is successfully carried out in FLUENT v13.0. And conclude that:

- The sloshing force fluctuation is more at the lower fill level and high acceleration.
- Increase in filling level in the tank, it increases the magnitude of the forces and moment due to the large mass of fuel but the variation of force is less.
- Transverse baffle with holes reduces slosh forces and moment significantly in the pitch plane which improves braking performance of the vehicle.
- Transverse baffle and transverse baffle with holes is not capable of reducing roll moment. And hence no effect on steering performance.
- Baffles are used as an anti-sloshing device in tank.

7.2 Scope for future work

This work leaves a wide scope for future investigators to explore many other aspects of sloshing. Some recommendations for future research include:

- The present study concerned with finding more effective type baffle under linear acceleration field. The study can be extended by considering different types of baffles like helical baffles with holes, porous baffle etc.
- Present work can be further extended to sloshing in nuclear reactor during earthquake.

******

REFERENCES

[1] K.M.Tehrani, S.Rakheja, I.Stiharu. "Three-dimensional analysis of transient slosh within a partly-filled tank equipped with baffles". Journal of vehicle system dynamics, vol. 45, pp.525-548.

[2] S.Rakheja, T.kandasamy, A.K.W. Ahemad. "An analysis of baffle design for limiting fluid slosh in partly filled tank trucks." The open transportation journal, 2010, pp.23-32.

[3] Raouf A. Ibrahim. "Liquid Sloshing Dynamics: Theory and Applications." Cambridge University Press, Cambridge, UK (2005).

[4] Craig K. J., Kingsley T. C., "Design optimization of container for sloshing and Impact". University of Pretoria, 33 (2007): pp.71-87.

[5] Takabatake D., Sawada S., Yoneyama N. and Miura M., "Sloshing reduction effect of splitting wall in cylindrical tank "The 14th World Conference on Earthquake Engineering" October, 12-17 (2008), Beijing, China.

[6] Eswaran M., Saha U.K., Maity D., "Effect of baffles on a partially filled cubic tank: Numerical simulation and experimental validation". Journal of computer and structures, 87(2009): pp.198-205.

[7] Chen Wei, Haroun Medhat A., Liu Feng, "Large amplitude liquid sloshing in a seismically excited tank." Earthquake Engineering and structural dynamics, 25 (1996): pp.653-669.

[8] Djavareshkian M.H. and Khalili M., "Simulation of Sloshing with the Volume of Fluid Method." Journal of fluid dynamics and material processing, 2 (2006): pp.299-307.

[9] Sakai F., Nishimura M., Ogawa H., "Sloshing behaviour of floating roof oil storage tanks." Journal of composites and structures, 19 (1984): pp.183-192.

[10] Pal N.C., Bhattacharyya S.K., Sinha P.K., "Non-linear coupled slosh dynamics of liquid filled laminated composite containers: a two dimensional finite element approach". *Journal of sound and vibration*, 261 (2002): pp.729-749.

[11] Abramson H.N., "The dynamic behaviour of liquids in moving containers". NASA SP-106, (1996): National Aeronautics and Space Administration. Washington, DC.

[12] Hasheminejad Seyyed M., Aghabeigi Mostafa, "Liquid sloshing in a half-full horizontal elliptical tanks", *Journal of sound and vibration*, 324 (2009): pp.332-349.

[13] Biswal K.C., Bhattacharyya S.K., Sinha P.K., “Dynamic response analysis of a liquid filled cylindrical tank with annular baffle”, *Journal of Sound and Vibration*, 274 (2004): pp.13-37

[14] Celebi, M.S., akyildiz, H., “Non-linear modeling of liquid sloshing in a moving rectangular tank”, Ocean Engineering, 29 (2002): pp.1527-1553.

[15] Rebouillat S., Liksonov D., “Fluid–structure interaction in partially filled liquid containers: A comparative review of numerical approaches” *Journal of Computers & Fluids*39 (2010): pp.739–746

[16] University of Washington, School of Oceanography. Linear wave theory lecture notes. Seattle, USA.

[17] Chen B.F, Chiang H-W. “Complete 2D and fully nonlinear analysis of ideal fluid in tanks.” *Journal of Ocean Engineering Mechanics* (ASCE) 1999; 125(1): pp.70-78.

[18] ANSYS-FLUENT 13.0 theory Guide, User guide, ANSYS Inc.

[19] Wang C.Z. and Khoo B.C. “Finite element analysis of two-dimensional nonlinear sloshing problems in random excitation.” *Journal of Ocean Engineering* 2005; 32: pp.107-133.

[20] Graham E W. “The forces produced by fuel oscillation in a rectangular tank.” *Douglas Aircraft Company Report* 1951; SM-13748.

[21] Fernando Meseguer-Garrido.“On the sloshing of liquids in parallelepiped-shaped containers.” European Journal of Physics 2003; 24:pp.277-288.

[22] Zhang J. “Non-linear wave theory course notes.” http://ceprofs.tamu.edu/jzhang/ Texas A&M University. Texas, USA 2003

[23] B. E. Launder and D. B. Spalding. “Lectures in Mathematical Models of Turbulence.” *Academic Press*, London, England, 1972.

[24] Y.M. Shen, C.O. Ng, Y.H. Zheng, “Simulation of wave propagation over a submerged bar using the VOF method with a two-equation k–epsilon turbulence modeling” Journal of Ocean Engineering 2004; 31:87–95

[25] Hirt C W, Nichols B D. “Volume of Fluid (VOF) method for the dynamics of free boundaries.” *Journal of Computational Physics*, 1981; **39**:201-225.

[26] Hadzic I, Mallon F, Peric M. “Numerical Simulation of Sloshing.” Fluent.In, Application Brief EX161,Tokyo, Japan 2001.

[27] Ibrahim, R. A., Pilipchuk, V. N. and Ikeda, T. “Recent Advances in Liquid Sloshing Dynamics,” Journal of Applied Mechanics 2001; 54:133–199.

[28] Ranganathan, R. and Yang,Y.S., 1996, Impact of liquid load shift on the braking characteristics of partially filledtank vehicles.Vehicle Systems Dynamics,23(3), 223–240.

[29] Strandberg, L., 1978, Lateral stability of road containers, Report No. 138A, VTI (The Swedish National Road and Traffic Research Institute), Sweden.

[30] Popov, G., Sankar, S., Sankar, T.S. and Vatistas, G.H., 1993, Dynamics of liquid sloshing in horizontal cylindrical road containers. Proceedings of Institution of Mechanical Engineers (UK), Part C: Journal of Mechanical Engineering Science, 207, pp.399–406.

[31] Wang, Zhanqi and Rakheja, S., 1995, Influence of partition location on the braking performance of a partiallyfilled tank truck.Transactions of SAE, Journal of Commercial Vehicles,104(2), 592–601.

[32] Ranganathan, R., Rakheja, S. and Sankar, S., 1990, Influence of liquid load shift on the dynamic response of articulated tank vehicles.Vehicle Systems Dynamics,19, 177–200.

[33] Popov, G., Sankar, S. and Sankar, T.S., 1993, Dynamics of liquid sloshing in baffled and compartmented road containers.Journal of Fluids and Structures,7, 803–821.

[34] Parizi, H., Pasandideh-Fard, M. and Dolatabadi, A., A Computer code for modeling sloshing, Proceeding of the 11th Annual Conference of the CFD Society of Canada, May 2003, Vancouver, BC, Canada, pp. 264–271.

[35] Sheu, T.W.H. and Lee, S-M., 1998, Large-amplitude sloshing in an oil tanker with baffle-plat/drilled holes, International Journal of Computational Fluid Dynamics,10, pp.45–60